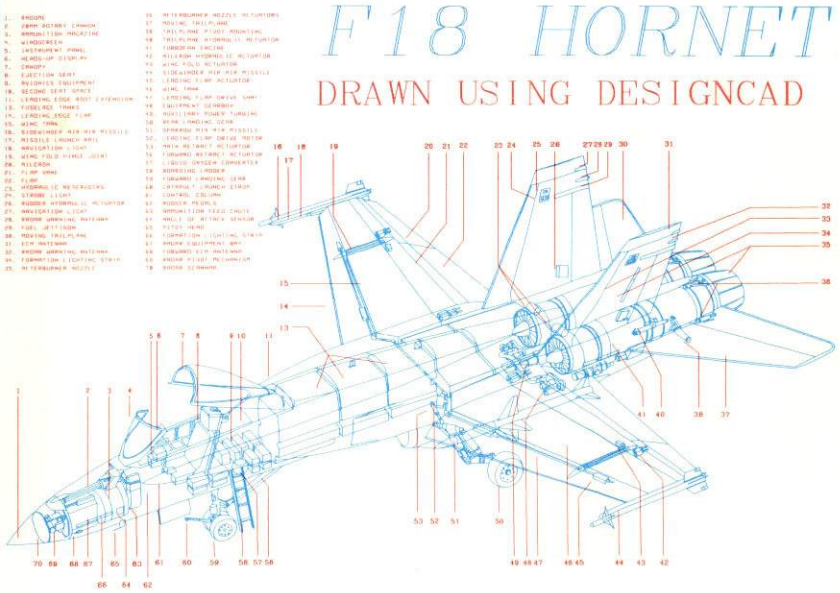
DesignCAD

The Easy to Use CAD System

Reference



DesignCAD

The Easy to Use CAD System

Version 3.0

Reference Manual

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GETTING STARTED - USING A HARD DRIVE

The first thing you should do is fill out your owner registration card and mail it. This will insure that you will be notified when future updates are available.

Then make a backup copy of your DesignCAD disks. Use the DISKCOPY command from DOS:

```
DISKCOPY
(or DISKCOPY A: B: if you have two floppy drives)
```

When instructed, place the first DesignCAD disk (source) and a blank disk (destination) into the floppy drive(s). After the copy operation is complete, make a copy of the other DesignCAD disks in the same manner. Store the original DesignCAD disks in a safe place.

Now create a DesignCAD subdirectory on your hard drive:

```
MD \DC
```

Next, change to the DesignCAD subdirectory and copy all the DesignCAD files to the hard drive using the following commands:

```
CD \DC
COPY A:*.*
```

Repeat the COPY command to copy files onto your hard drive from EITHER Program Disk #1 or #2, but not both. Use Program Disk #1 if you DO NOT have a math coprocessor, and Program Disk #2 if you DO have a math coprocessor. Then copy all the other DesignCAD disks.

NOTE: DO NOT COPY PROGRAM DISK #2 ONTO YOUR HARD DRIVE IF YOU DO NOT HAVE A MATH COPROCESSOR.

Step 1: Make a DesignCAD Subdirectory.
Step 2: Change to the DesignCAD Subdirectory.
Step 3: Copy Program Disks #1 OR #2, and all the other DesignCAD disks. (Program Disk #1 is for use without a math coprocessor, Program Disk #2 is for use with a coprocessor.
Step 4: Run SETUP.
Step 5: Enter "DC" to run DesignCAD.

NOTE: DO NOT COPY PROGRAM DISK #2 ONTO YOUR HARD DRIVE IF YOU DO NOT HAVE A MATH COPROCESSOR.

GETTING STARTED - USING 5.25" FLOPPY DISK DRIVES

The first thing you should do is fill out your owner registration card and mail it. This will insure that you will be notified when future updates are available.

Then make a backup copy of your DesignCAD disks. Use the DISKCOPY command from DOS:

```
DISKCOPY
(or DISKCOPY A: B: if you have two floppy drives)
```

When instructed, place the first DesignCAD disk (source) and a blank disk (destination) into the floppy drive(s). After the copy operation is complete, make a copy of the other DesignCAD disks in the same manner. Store the original DesignCAD disks in a safe place.

If you do not have a math coprocessor, place the Print disk into drive A: and enter ALTPRINT to select the correct print program. (To change back to the coprocessor version of the Print program, enter ALTPRINT 87.)

You are now ready to run the DesignCAD Configuration program. Place the CONFIGURATION DISK into drive A:, and enter SETUP. Enter all the configuration information, (See the following section) and insert the PRINT DISK and the correct PROGRAM DISK into drive A: when instructed. (Use Program Disk #1 if you DO NOT HAVE a math coprocessor, and Program Disk #2 if you DO HAVE a coprocessor.)

Then, enter DC to run DesignCAD.

Summary:

Step 1: Backup the original DesignCAD disks.

Step 2: If you do not have a math coprocessor, place the Print Disk in drive A: and enter ALTPRINT.

Step 3: Place the CONFIGURATION DISK into drive A: and run SETUP.

Step 4: Replace the disks as instructed

Step 5: Enter "DC" to run DesignCAD.

GETTING STARTED - USING 3.5" FLOPPY DISK DRIVES

NOTE: DesignCAD is available on 5.25" and 3.5" disks. If you have the wrong disk size, you can send the package to us for a free exchange:

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Begin by making a backup copy of your DesignCAD disks. Use the DISKCOPY command from DOS:

DISKCOPY A: B:

When instructed, place the DesignCAD program disk (source) into drive A and a blank disk (destination) into drive B. After the copy operation is complete, make a copy of the remaining disks in the same manner. Store the original DesignCAD disks in a safe place.

You are now ready to run the DesignCAD setup program. Place the CONFIGURATION DISK into drive A:, and enter SETUP. Enter all the configuration information, (See the following section).

When you are asked to insert the Print Disk into drive A:, insert the Program Disk (#1 if you DO NOT HAVE a math coprocessor, or #2 if you DO HAVE a math coprocessor). Leave that disk in and press ENTER when you are asked to insert the Program Disk.

Then, enter DC to run DesignCAD.

Summary:

Step 1: Backup the original DesignCAD disks.

Step 2: Place the CONFIGURATION DISK into drive A: and run SETUP.

Step 3: When instructed, replace the Configuration Disk with the Print Disk in the floppy disk drive.

Step 4: Insert Program Disk #1 (for use without a math coprocessor) or Program Disk #2 (for use with a coprocessor) into drive A: when instructed. (Use the Program Disk in place of the Print Disk, also.)

Step 5: Enter "DC" to run DesignCAD.

THE CONFIGURATION PROGRAM

Most printers, plotters, and digitizing pads have their own distinct control codes used to perform their various functions. Because of this, you need to "tell" DesignCAD which devices you use. To do this, run the program SETUP (Enter SETUP from the A> or the C> prompt).

If you do not have a hard drive, you will need to start the SETUP program with DesignCAD Configuration Disk in the floppy drive, and then replace the Configuration Disk with the DesignCAD Print Disk and Program Disk. (Use Program Disk #1 if you do not have a math coprocessor, and Program Disk #2 if you do have a coprocessor.)

First, you will be asked whether you want to use English or Metric units of measurement. These are the units of measurement for the plotter, printer, and digitizer sizes. Your drawings can always be in either English or Metric units.

Next, you will be asked what type of printer you have. If your printer does not appear on the page, press ENTER to go to the next page. To return to the previous page, press ESC. There are several pages of printers.

After you select the printer, you will be asked to select the printer output device - LPT1:, LPT2:, COM1:, or COM2:. Select the port to which your printer is connected. (LPT1: is the most common printer port.)

Next, you will be asked for the type of plotter. If your plotter is not listed among those on the screen, you may select "NONE OF THESE" to set up your own custom plotter codes. As with the printers, press ENTER to go to the next page in the list of plotters, or ESC to go to the previous page.

There are five possible plotter output devices:

1. LPT1:
2. LPT2:
3. COM1:
4. COM2:
5. Disk File

Select the output port to which the plotter is connected. If you need to output the plotter codes to a disk file instead of directly to the plotter, select option 5. When a disk file is used for the plotter output device, you will be asked for the output file name when each plot is made.

After entering the plotter information, you will be asked for the type of mouse or digitizing tablet you use. (Digitizing tablets are also called digitizers and graphics pads.) If you have a digitizing tablet, you will be asked for the size of the tablet and the serial port information after you select the tablet. Be sure the serial information you enter corresponds to that of your digitizing tablet.

If you have a mouse, the mouse device driver supplied by the manufacturer must be loaded before the mouse is used by DesignCAD. (Use MOUSE.COM for the Microsoft or Logitech Mouse, MSMOUSE.COM for the Mouse Systems Mouse).

Next, you will be asked which type of display / graphics adapter you use. If you are not sure, choose the first or second option, IBM Standard Color Graphics. If you are using a monochrome monitor with a 9-pin connector and you aren't sure about the display type, try the Hercules Monochrome option.

NOTE: Be sure that you have selected the correct graphics display. If you find that DesignCAD 3-D displays only the copyright notice during initial

startup and will not proceed, you either have selected the wrong graphics adapter in the SETUP program or the graphics card and/or computer is improperly configured.

After you select the display, you will be asked if you want to change the screen aspect ratio. On your computer display, the horizontal dots or pixels are not the same distance apart as the vertical pixels. The aspect ratio is the distance between horizontal pixels divided by the distance between vertical pixels. The default value should be correct for your screen.

However, if you find that circles are too tall on your screen, make the aspect ratio smaller. Similarly, if a circle is flat on the screen, the aspect ratio can be increased in value. The aspect ratio does not affect the way drawings are printed or plotted.

After all the setup information is entered, the peripheral devices are displayed on the screen. You can change any of them at this point. When all the information is correct, select option 8 to save the information to disk.

If you have a floppy disk drive system, you will be asked to insert different disks into the drive to save the configuration information on the other disks.

If you use a digitizing tablet, you should align the digitizer menu. To do this, select option 6 in the Setup program.

The digitizer menu is the list of commands on a rectangular card. This can be affixed to the upper left corner of your digitizing pad to allow command entry from the digitizing pad.

If you have problems using a mouse, digitizing tablet, or plotter, see the list of cable configurations and switch settings in the DesignCAD 3-D Reference Manual.

After the Setup program has been run once, you can run it again and change only the configuration items than need to be changed. For example, if you need to set the program for a different plotter, you can select just the plotter option. You will not have to enter the information on printers, digitizers, etc.

Now you are ready to load DesignCAD. Enter:

DC

After a brief copyright notice, you will see a screen like one in Figure 1. There is a box framing most of the screen and a status line above. In the center of the box is the cursor. You will use the cursor for entering points on the screen for commands.

To the right of the drawing screen is the command menu. (This can be turned off with the System Parameter command to allow a larger drawing area.) Directly beneath the drawing screen is the Command Line. This is the line from which information is entered for the DesignCAD commands. It is normally blank. To exit DesignCAD, press the F8 key.

NOTE: DesignCAD uses most of the memory in your computer. If you are using any RAM-resident software, it may conflict with DesignCAD. DesignCAD uses MS-DOS standards to allocate memory. Any RAM-resident software that does not can causes unpredictable results.

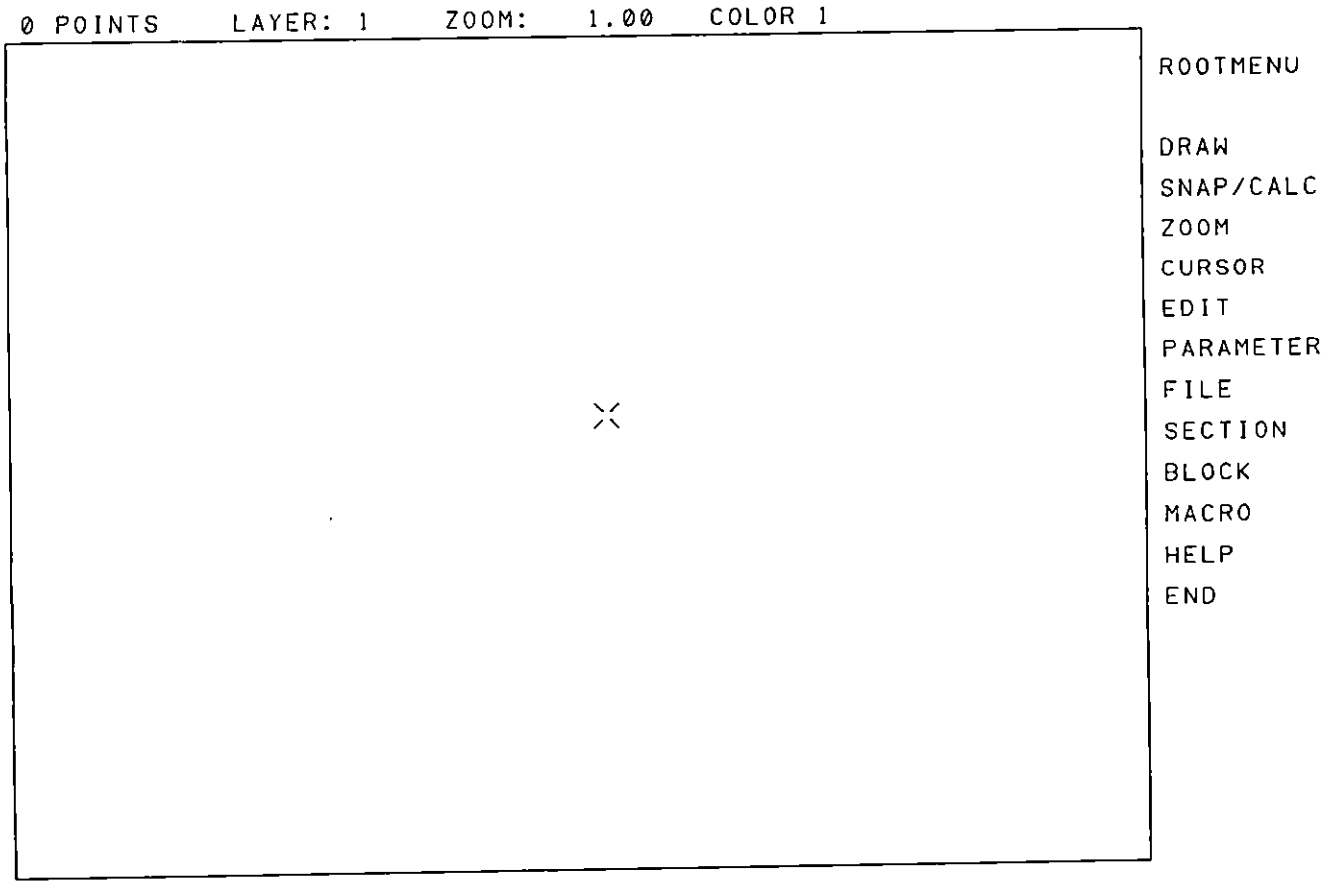
0 POINTS LAYER: 1 ZOOM: 1.00 COLOR 1
ROOTMENU
DRAW
SNAP/CALC
ZOOM
CURSOR
EDIT
PARAMETER
FILE
SECTION
BLOCK
MACRO
HELP
END

FIGURE 1

CONTROLLING THE CURSOR

You can move the cursor with the numeric keypad using numbers 7, 8, 9, 4, 6, 1, 2, and 3, as shown in Figure 2. Try it. The cursor moves in the direction of the number on the keypad. If the numeric lock (or shift key) is on, the cursor moves in small steps. If the numeric lock is off, the cursor moves in larger steps. It is convenient to leave the numeric lock off for normal cursor movement, and press the shift key for precise cursor positioning.

There are also cursor control keys to move the cursor to the top, bottom, left, or right edge of the screen:

CTRL-4 - Cursor to Left Edge of Screen
CTRL-6 - Cursor to Right Edge of Screen
CTRL-7 - Cursor to Top of Screen
CTRL-1 - Cursor to Bottom of Screen

"CTRL-" means to hold the CTRL key down when you press the 4, 6, 7, or 1 key. (These cursor controls function only on the numeric keypad.)

DesignCAD also supports a mouse or digitizing tablet for cursor movement. This manual refers to cursor movement using the keyboard. If you have a mouse or digitizing tablet, the cursor may be moved using either the keyboard or digitizer.

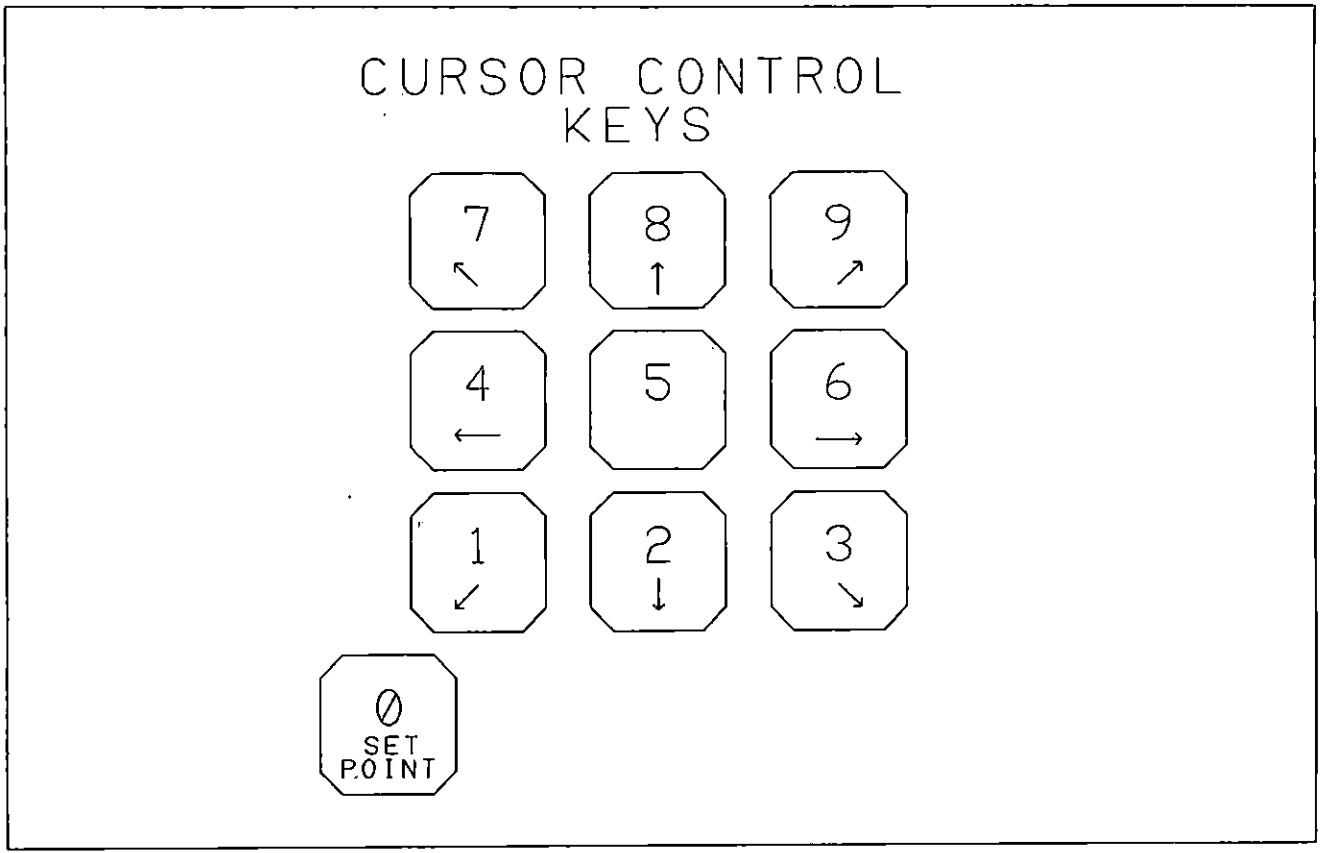

FIGURE 2

DRAWING A LINE

Most drawing commands require a set of points for reference. To set a point, move the cursor to the desired position and press the 0 key on the numeric keypad. Try setting three points in a triangle. Next, press the V key (For Vector). DesignCAD will draw lines from one point to the next. Press the ESC key when you finish. The ESC key is used to "escape from" or undo the last line or entity drawn.

The Line command is one of many DesignCAD commands. A later section explains each of these commands in detail.

WRITING YOUR NAME

DesignCAD has excellent text display capabilities. For example, set one point near (about an inch away from) the lower left corner of the screen, as shown in Figure 3. Set another point to the right, near the lower right corner of the screen. Next press the T key (for Text). You will be asked to enter "TEXT:" at the top of the screen. Type your name, followed by the ENTER key.

Your name will be displayed between the two points. You can position the two points anywhere on the screen to display text at any location, size, or angle.

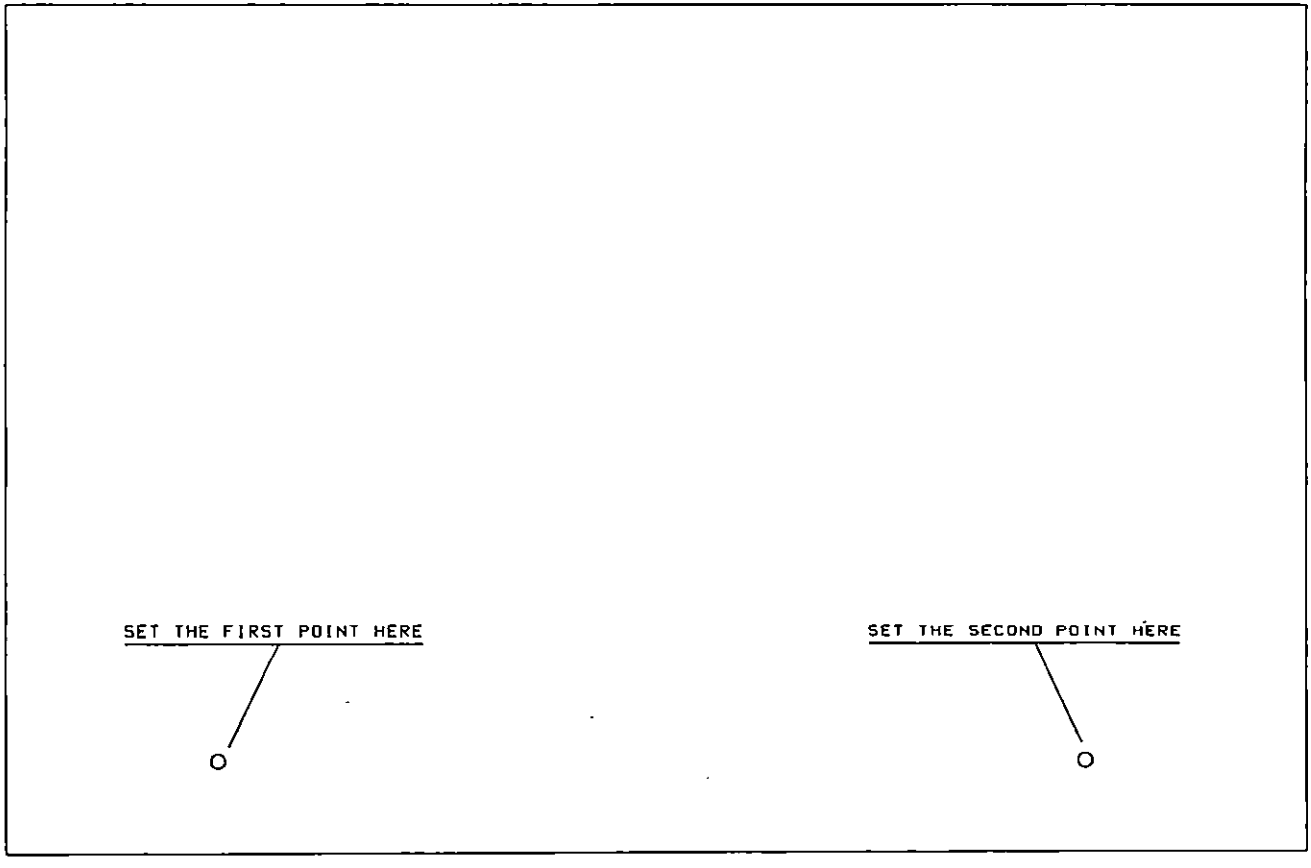

FIGURE 3

USING THE ZOOM

With your name still on the screen, move the cursor to the first letter of your name. Set a point in the center of that letter. Then press the Z key. When asked for the Zoom Factor, enter 2. The first letter of your name will appear on the screen twice as large as it was originally. The rest of your name will follow the first letter, although some of it will lie off to the right of the screen.

THE VIRTUAL SCREEN

The Virtual screen is used with the standard resolution IBM Color Graphics display (option 1 or 2 on the SETUP display selection). If you have a high resolution display, the entire drawing screen can be displayed on your monitor and the virtual screen is not needed. The virtual screen is used only with the standard IBM Color Graphics display, so if you have a high resolution display you may skip this section.

The standard IBM Color Graphics display is somewhat limited in size and resolution. DesignCAD allows you to get around this problem by providing the ability to draw on a virtual screen four times larger than your physical display screen. The physical screen displays one quarter of the virtual screen at any given time. It can be thought of as a "window" looking onto the virtual screen.

The window can be moved about so that you can see any portion of the virtual screen. The function keys F1 through F4 are used to move the "screen window".

At the right of the status line is a rectangle showing which portion of the virtual screen is currently being displayed.

After you zoomed in on the first letter of your name, the rest of your name could not be read because it was off to the right of the screen. Use the F2 key (Screen Window Right) to observe the part of the virtual screen to the right. Use the F1 key (Screen Window Left) to return to the original position.

Now try out the Screen Window Up (F3) and Screen Window Down (F4) keys.

Throughout this manual, the screen image and the virtual screen will be referred to. The screen image is the portion of the virtual screen that appears on

your display. The virtual screen contains four times the area shown on your display.

DESIGNCAD COMMANDS

Many of the DesignCAD commands are Keystroke commands, that is, commands that are executed by pressing only one key on the keyboard. These commands are the most commonly used DesignCAD commands.

Other DesignCAD commands can be entered from the Command Menu or Command Line. These are called the "menu commands".

A DesignCAD menu command is entered by pressing the ENTER key and then selecting the appropriate item from the Menu. (The Command Menu is the list of commands to the right of the drawing screen.) A menu command can also be entered by pressing the ENTER key and then typing the menu command name on the command line.

The quick reference card and the on-screen help (? key) can help you get used to the new command language. The DesignCAD commands are listed on the following pages.

DesignCAD Commands

Arc	Draw an arc - center, radius, angle.
Arc-2	Draw an arc - beginning, end, radius.
Arc-3	Draw an arc through 3 points.
Area	Calculate area.
Arrow	Draw an arrow.
Attribute	Enter an Attribute.
Block Add	Add an entity to a Block.
Block Circular Repeat	Duplicate Block in a circle.
Block Color	Change the color of a Block.
Block Delete	Delete a Block.
Block Handle	Set or move the Block's Handle.
Block Insert	Copy a Block one time.
Block Layer	Change the layer of a Block.
Block Mirror	Make a mirrored copy of a Block.
Block Move	Move a Block from one location to another.
Block Repeat	Duplicate a Block a number of times.
Block Reset	"Undefine" a Block.
Block Rotate	Rotate a Block.
Block Save	Save a Block to disk.
Block Set	Define a Block.
Block Set, cutoff	Define Block using Line Breaking.
Box	Draw a box.
Chamfer	Chamfer a corner.
Change	Change the characteristics of a line.
Change Color	Change the color of one or more lines
Change Directory	Change to another subdirectory.
Circle	Draw a circle using center and radius.
Circle-2	Draw a circle using the diameter.
Circle-3	Draw a 3-point circle.
Clear All	Clear the Screen.
Color	Change drawing color.
Cursor Type	Change cursor type.
Curve	Draw a curve.
Cutoff	Break lines around a section.

DesignCAD Commands (continued)

Dimension	Dimension horizontally or vertically.
Dimension Line	Dimension at any angle.
Dimension Angle	Dimension an angle.
Directory	Display drawings on disk.
Drawing Parameter	Enter Drawing Parameters.
Ellipse	Draw an ellipse or Oval.
Elliptical Arc	Draw an elliptical arc.
End	Stop the program.
Erase	Erase an entity.
Fillet	Fillet a corner.
Gravity Move	Move the cursor to the nearest point.
Gravity Point	Move the cursor to the nearest point and set a point there.
Grid	Turn the Display Grid on or off.
Hatch	Fill an area with a pattern.
Hatch Line	Fill the area within a line.
Help	Display DesignCAD menu commands.
ID	Display information about a line.
Intersect	Snap to the nearest intersection.
Join	Connect endpoints of two or more lines.
Layer	Layer command.
Line	Draw a line.
Line Type	Select the line type.
Line Snap	Snap to the nearest line.
Macro Define	Define a Macro.
Macro Execute	Execute a Macro.
Macro Save	Save a Macro being defined.
Midpoint	Snap to the midpoint of a line.
Oops	"Unerase" the last thing erased.
Origin	Move the origin.
Ortho Line	Draw a line vertically or horizontally.
Overlay	Read an overlay from disk.
Paint	Fill an area.
Pan	Pan the drawing across the screen.
Parallel	Draw a parallel line.
Perpendicular	Draw a perpendicular line.
Plot	Output the drawing to the plotter.

DesignCAD Commands (continued)

Point Move	Move a point in an entity.
Point Set	Set a point at the current cursor position.
Point Set, Polar	Set a point using polar coordinates.
Point Set, Relative	Set a point using relative X-Y coordinates
Point Set, Absolute	Set a point using absolute X-Y coordinates
Polygon	Draw a regular polygon.
Print	Print drawings.
Quarter Circle	Draw a quarter circle.
Repeat Section Copy	Repeat section copy.
Retrieve	Read a drawing or symbol from disk.
Rotate	Rotate the drawing.
Rubber Band Box	Enable the rubber band box.
Rubber Band Line	Enable the rubber band line.
Save	Save the drawing.
Section Copy	Copy a Section.
Section Copy, cutoff	Copy a Section, with cutoff
Section Delete	Delete a Section.
Section Delete, cutoff	Delete a Section, with cutoff.
Section Move	Move a Section.
Section Move, cutoff	Move Section, with cutoff.
Section Save	Save a Section.
Section Save, cutoff	Save a Section with cutoff.
Semi-Circle	Draw a Semicircle
Sketch	Enable the sketch mode.
Snap Grid	Turn the Snap Grid on or off.
Status Line Format	Set the status line format.
Stretch	Stretch a Section.
System Parameter	Enter System Parameters.

DesignCAD Commands (continued)

Tangent	Draw a line tangent to 1 circle.
Tan-2	Draw a line tangent to 2 circles.
Text	Draw Text.
Text Arc	Draw Text in an arc.
Trim	Trim a line (shorten or lengthen).
Undo	Undo the last line or entity drawn.
Units	Define units of measurement.
Vector Arc	Draw Elliptical Arc made of vectors.
Vector Circle	Draw a Circle made of vectors
Vector Curve	Draw a Curve made of vectors.
Vector Ellipse	Draw an Ellipse made of vectors.
Vector Text	Draw Text made of vectors.
Width	Enter the line width or thickness.
Zoom	Zoom in or out.
Zoom Window	Zoom with a window.
Zoom Static	Static Zoom

Command	Menu Command	Short Form	Keystroke Command	Page
Arc	ARC			36
Arc-2	ARC-2	A2		37
Arc-3	ARC-3	A3		38
Area	AREA			40
Arrow	ARROW		>	41
Attribute	ATTRIBUTE	ATT	$	42
Block Add	BLK ADD	BA		44
Block Circular Repeat	BLK CIRC	BC		45
Block Color	BLK COLOR	BCO		47
Block Delete	BLK DEL	BD		48
Block Handle	BLK HANDL	BH		49
Block Insert	BLK INSRT	BI		50
Block Layer	BLK LAYER	BL		52
Block Mirror	BLK MIR	BMI		53
Block Move	BLK MOVE	BM		54
Block Repeat	BLK REPT	BR		55
Block Reset	BLK RESET	BRE		57
Block Rotate	BLK ROTAT	BRO		58
Block Save	BLK SAVE	BS		59
Block Set	BLK SET	B		60
Block Set, Cutoff			ALT-B	60
Box	BOX		]	62
Chamfer	CHAMFER	CHF	ALT-F	63
Change	CHANGE	CH		64
Change Color			ALT-H	65
Change Directory	CHDIR	CD		66
Circle	CIRCLE	CIR	O	67
Circle-2	CIRCLE-2	C2		69
Circle-3	CIRCLE-3	C3		70
Clear All	CLEAR ALL	CLS	Y	71
Color	COLOR		H	72
Cursor Type	CURS TYPE	CT	+	73
Curve	CURVE		C	74
Cutoff	CUTOFF	CUT		76

Command	Menu Command	Short Form	Keystroke Command	Page
Dimension	DIMENSION	DIM	@	77
Dimension Line	DIM LINE	DL		79
Dimension Angle	DIM ANGLE	DA		81
Directory	DIRECTORY	DIR	F7	82
Drawing Parameter	DRAWING	DP	Q	83
Elliptical Arc	ELLIP ARC	EA	A	91
Ellipse	ELLIPSE		O	94
End	END		F8	97
Erase	ERASE	E	E	98
Fillet	FILLET	F	F	99
Gravity Move	GRAV MOV	GM	,	101
Gravity Point	GRAVITY	GV	.	102
Grid	GRID		G	103
Hatch	HATCH	HA	#	104
Hatch Line	HATCHL	HL		106
Help	HELP		?	108
ID	ID		I	109
Intersect	INTERSECT	INT		110
Join	JOIN			111
Layer	LAYER	LAY	L	113
Line	LINE		V	116
Line Type	LINE TYPE	LT	/	118
Line Snap	LINESNAP	LS	K	119
Macro Define	MAC DEF	MD		120
Macro Execute	MAC EXEC	EXEC		122
Macro Save			&	120
Midpoint	MIDPOINT	MID		123
Oops	OOPS		!	124
Origin	ORIGIN			125
Ortho Line	ORTHO LIN	OL	ALT-V	126
Overlay	OVERLAY			127
Paint	PAINT		P	129
Pan	PAN			131
Parallel	PARALLEL	PARA	=	132
Perpendicular	PERP			134

Command	Menu Command	Short Form	Keystroke Command	Page
Plot	PLOT		F6	135
Point Move	POINT MOV	PM	*	137
Point Set			0	139
Point Set, Polar	POINT PLR	PP	;	140
Point Set, Relative	POINT REL	PR		141
Point Set, Absolute	POINT XY	PXY	:	143
Polygon	POLYGON	POLY		144
Print	PRINT		F5	145
Quarter Circle	QUART CIR	QC	(	148
Repeat Section Copy	REPT COPY	RC	"	149
Retrieve	RETRIEVE	LOAD	F9	150
Rotate	ROTATE	R	R	154
Rubber Band Box	RUB BOX	RB	[	155
Rubber Band Line	RUB LINE	RL	-	156
Save	SAVE		F10	157
Section Copy	SEC COPY		N	160
Section Copy, Cutoff			ALT-N	160
Section Delete	SEC DEL		D	163
Section Delete, Cutoff			ALT-D	163
Section Move	SEC MOVE		M	165
Section Move, Cutoff			ALT-M	165
Section Save	SEC SAVE		X	167
Section Save, Cutoff			ALT-X	167
Semi-Circle	SEMI CIRC	SC	)	169
Sketch	SKETCH			170
Snap Grid	SNAP GRID	SN	ALT-G	171
Status Line Format	STATUS LN	SL	J	172
Stretch	STRETCH	S	S	174
System Parameter	SYSTEM	SP	ALT-Q	176
Tangent	TANGENT	TAN		184
Tan-2	TAN-2	TAN2		185
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DESIGNCAD COMMAND REFERENCE

Each of the DesignCAD commands can be executed using the menu or the command line.

To use a command from the menu, press ENTER to move the cursor to the menu, and use the arrow keys to move about the menu. Select a command by pressing ENTER.

To enter a command on the command line, press enter and type the name of the command. The short form of the command can be used, such as PS instead of PLANESNAP.

If you are using a mouse, you can enter the command mode by moving your mouse all the way to the right. You can then make your selection by moving the mouse up or down and pressing the first button on the mouse at the desired command.

Most of the selections on the first menu, or Root Menu, will take you to a sub-menu. For example, to draw a curve, select DRAW from the Root Menu. Then select CURVE from the next menu.

To return to the Root Menu, press HOME, or select ROOT MENU at the bottom of the menu. The Page-Up key can be used to move back to the previous menu.

The Menu commands can also be entered from the command line. To do this, enter the command mode by pressing ENTER. Then, type the name of the command and press ENTER. That command will then be performed.

Many of the menu commands have short forms which can be used when entering the commands from the command line. For example, C3 can be used instead of CIRCLE-3 to draw a 3-point circle. In addition, those menu commands with corresponding keystroke commands can be

performed by entering the keystroke command on the command line.

Many of the DesignCAD commands also have corresponding Keystroke commands, that is, commands that are executed by pressing only one key on the keyboard. These commands are the most commonly used DesignCAD commands. To use a DesignCAD keystroke command, press that key. For example, press V to draw a line through a set of points. You do not have to press enter before you press the key for a keystroke command.

When you use the menu or enter commands from the command line, you can enter the required points before or after entering the command. When you use a keystroke command, you must set all required points before pressing the key for the command.

For example, you can draw a line by entering LINE from the command line. You will be asked to set the points through which lines are to be drawn, if you have not done so. If you use the keystroke command "V", those points must be set before pressing "V".

When the phrase "enter ARC" or "enter <command>" is used in the following section, it means to enter the command either from the menu, the command line, or to press the single key for the keystroke command.

When you enter the command from the command line, the short form of the command can be used (A3 instead of ARC-3, for example).

When you use the keystroke commands, remember to set all required points before pressing the key.

ARC - Draw an Arc using radius and angle.

Menu Command: ARC
Keystroke Command: A
Keystroke Command: ALT-A

Point 1: Center of arc.
Point 2: Beginning of arc.

This command draws a circular arc. Two points are required:

1. A point at the center of the arc.
2. A point representing the radius of the arc.

The distance from the first point to the second is the radius of the arc. Next, you will be asked for the angle of the arc. An arc will then be drawn from the first point counter-clockwise for a distance of that angle. (The angle is relative to the second point.) See Figure 4.

The ALT-A keystroke command can be used to draw an arc made of short line segments, rather than an actual Arc entity. This is sometimes useful if the arc is to be slanted or distorted.

If the keystroke command A or ALT-A is used with more than two points set, an elliptical arc will be drawn. See the Elliptical Arc command.

ARC-2 - Draw an arc between two points.

Menu Command: ARC-2
Short Form: A2

Point 1: Beginning of arc.
Point 2: Ending of arc.

The Arc-2 command is used to draw an arc (a portion of a circle) between two points. To use the command, set 2 points:

1. The beginning point that lies on the arc.
2. The ending point that lies on the arc.

You will be asked for the length of the radius of the arc. An arc of that radius will then be drawn from the first point counter-clockwise to the second. See Figure 4.

ARC-3 - Draw a 3-point arc.

Menu Command: ARC-3
Short Form: A3

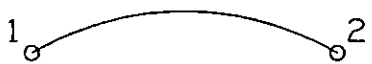
Point 1: Beginning of arc.
Point 2: Any point on the arc except endpoints.
Point 3: Ending of arc.

This command draws a circular arc through three points, passing through the first point, then the second, and then the third point. See Figure 4.

ARC COMMANDS

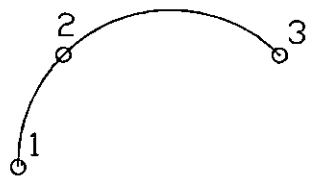
ARC

SET TWO POINTS
AND ENTER THE ANGLE



ARC-2

SET TWO POINTS
AND ENTER THE RADIUS



ARC-3

SET 3 POINTS

FIGURE 4

AREA - Calculate Area.

Menu Command: AREA

Points 1-N: Outline of the area to be calculated.

The area command is used to calculate the area of a part of the drawing. To use the command, enter AREA and set points around the outside of the area to be measured. The points will be connected with a temporary line, and the area of the shape will be calculated and displayed at the bottom of the screen.

The Drawing Units are the units of measurement for the area. (See Units command). If the first and last point are not in the same location, an imaginary line will connect those two points to close the shape.

The ID command can also be used to calculate area.

ARROW - Draw an Arrow.

Menu Command: ARROW
Keystroke Command: >

Point 1: Beginning Point
Point 2: Location for arrowhead.

This command draws an arrow from one point to another. Two points are used with the arrow command. An arrow is drawn from the first point to the second. The size of the arrowhead is determined by the default text size.

If more than two points are used with this command, a line is drawn through all the points and an arrow is drawn at the last point.

ATTRIBUTE - Enter an Attribute.

Menu Command: ATTRIBUTE
Short Form: ATTR
Keystroke Command: $

Point 1: Location for Attribute

The Attribute command is used to assign information to an object in the drawing. This information can later be extracted from the drawing file and used by other programs for applications such as parts lists and bills of materials.

To use the command, enter ATTRIBUTE and set a point at which the Attribute is to be located, or set the point and press "$". You will then be asked for the Attribute. Enter a line of text that will be associated with that location in the drawing.

Attributes will not be displayed on the screen unless the Display Attribute flag is enabled under the System Parameter command. Even though you cannot see an Attribute, it can be moved and copied along with the rest of the drawing.

This feature may be used with the DesignCAD Materials List program to provide a list of materials or other information in a drawing.

BLOCK COMMANDS

A Block is a set of lines or drawing entities that have been grouped together so they can be manipulated as a unit. There are DesignCAD commands for the definition of a Block, as well as copying, moving, etc.

A Block is defined by designating existing lines or drawing entities to be included in the Block. Once a Block has been defined, the Block commands will operate on that Block until it has been reset, erased, or replaced by another Block.

While a Block is similar to the DesignCAD Section, there are some important differences. The Section commands can be performed using keystroke commands, and are generally one-step commands. A Section is always defined as all the lines or entities within a "window". A Section can be copied, moved, deleted, and saved. A Section is always manipulated using the corners of its "window" for reference points.

A Block must be defined before it can be manipulated. A Block can be any subset of the drawing - not just those lines or entities in a window. With a block, you can define one or two reference points or "handles" for manipulation. There are many more operations that can be performed on a block than with a Section.

BLOCK ADD - Add a line or entity to a Block.

Menu Command: BLK ADD
Short Form: BA

Point 1: Entity to be added to block.

The Block Add command adds a line or entity to the current Block. To use this command, enter BLK ADD and set a point on the line or entity to be added to the Block.

BLOCK CIRCULAR REPEAT - Duplicate a Block in a
in a circle.

Menu Command: BLK CIRC
Short Form: BC

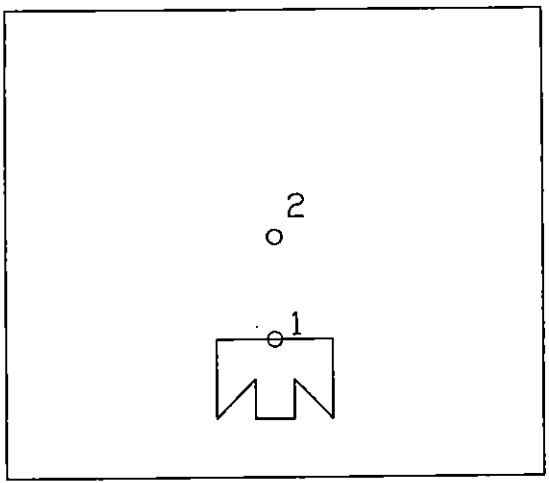
Point 1: Location for first block occurrence
Point 2: Center of rotation

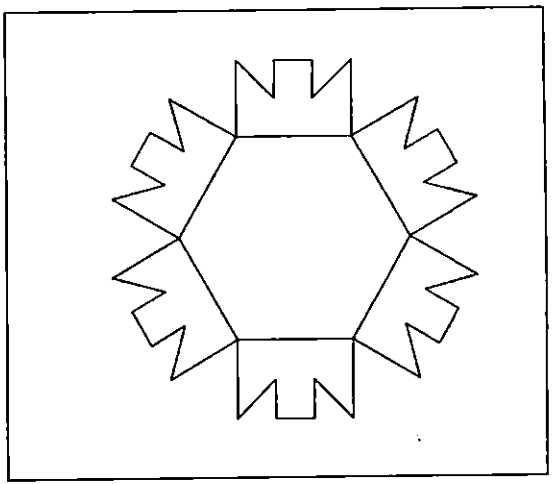
The Block Circle command can be used to duplicate a block several times around a circle or arc. Two points are used with the command - the first point is the location for the primary reference point of the Block, or handle (See the Block Handle command below.) The second point is the center of the circle about which the block is to be duplicated.

Next, the number of times the block is to be copied is entered, and the number of degrees in which these copies are to be placed. (Press ENTER for a full circle.) See Figure 5.

If the first copy lies directly on the original block, the block will not be duplicated onto itself. Otherwise, the first copy of the Block will be at the first point.

BLOCK CIRCULAR REPEAT





THE BLOCK CIRCULAR REPEAT COMMAND IS USED TO COPY A BLOCK IN A CIRCLE. THE FIRST POINT IS THE LOCATION FOR THE FIRST COPY. THE SECOND POINT IS THE CENTER OF THE CIRCLE ABOUT WHICH THE BLOCK IS COPIED.

FIGURE 5

BLOCK COLOR – Change the color of a Block.

Menu Command: BLK COLOR
Short Form: BCO

The Block Color command is used to change the color of all the lines or entities in a Block. After entering the new color for the Block, each entity in the block is redrawn in the new color.

BLOCK DELETE - Delete a Block.

Menu Command: BLK DEL
Short Form: BD

The Block Delete command erases or deletes from the drawing each line or entity in the Block. If this command is executed in error, the Oops command can be used to "unerase".

BLOCK HANDLE - Set or move the Block's handles.

Menu Command: BLK HANDL
Short Form: BH

Point 1: Primary Handle.
Point 2: Secondary Handle (Optional).

A Block can have one or two reference points, or handles. These handles are used with several of the Block commands as reference points for manipulating the Block.

The Block Handle command can be used to set the location of the handles. To use the command, enter BLK HANDL and set two points, one at the primary handle and one at the secondary handle.

If no handles are specified for a block, the lower left and lower right corners of the block are used. The handles can also be defined with the Block Set command (BLK SET, or the keystroke "B").

BLOCK INSERT - Copy a Block one time.

Menu Command: BLK INSRT
Short Form: BI

Point 1: Location for primary handle.
Point 2: Location for secondary handle.

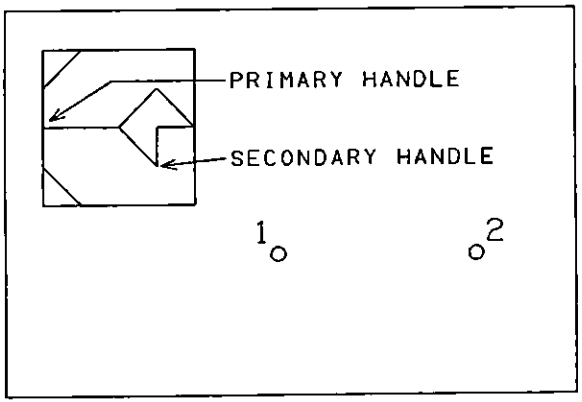
The Block Insert command makes a copy of the Block. One or two points may be used to specify the location at which the Block is to be copied. The primary handle of the Block is positioned at the first point.

If a second point is used, the Block's size and angle is adjusted so that the secondary handle is located at the second point. See Figure 6.

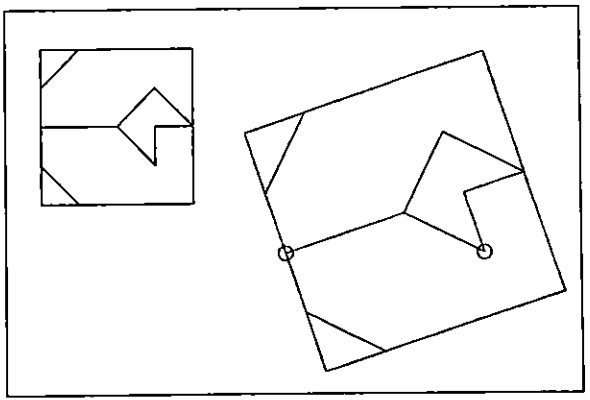
If no handles have been specified for the Block, the lower left and lower right corners of the block are used for handles.

This command is like the Block Move command, but it makes a copy of the block rather than moving the original block.

BLOCK INSERT



BEFORE



AFTER

THE BLOCK INSERT COMMAND MAKES A COPY OF A BLOCK, AND SCALES AND ROTATES THE BLOCK SO THAT THE HANDLES FIT ON THE POINTS SPECIFIED.

FIGURE 6

BLOCK LAYER - Change the layer of a Block.

Menu Command: BLK LAYER
Short Form: BL

The Block Layer command is used to change the layer of all the lines or entities in a Block. After the new layer number is entered, each entity in the block is assigned to the new layer.

BLOCK MIRROR - Make a mirror image of a Block.

Menu Command: BLK MIR
Short Form: BMI

Point 1: Location for "mirrored" block.

The Block Mirror command can be used to make a mirror image out of a block. To use the command, set a point at which the mirror image of the block is to be located. You will then be asked "Horizontal or Vertical?". If the mirror image is to be along the horizontal axis, then enter "H". Likewise, enter "V" for a vertical mirror image.

The Block Mirror command makes a copy of the block that is "mirrored" and locates it at the specified point.

BLOCK MOVE - Move a Block from one location to another.

Menu Command: BLK MOVE
Short Form: BM

Point 1: Location for primary handle.
Point 2: Location for secondary handle.

The Block Move command move a Block from one location to another. One or two points may be used to specify the location to which the Block is to be move. The primary handle of the Block is located at the first point.

If a second point is used, the Block's size and angle is adjusted so that the secondary handle is located at the second point.

If no handles have been specified for the Block, the lower left and lower right corners of the block are used for handles.

This command is like the Block Insert command, but it move the block rather than making a copy of it.

BLOCK REPEAT - Duplicate a Block a number of times.

Menu Command: BLK REPT
Short Form: BR

Point 1: Location for first copy.
Point 2: Location for second copy.

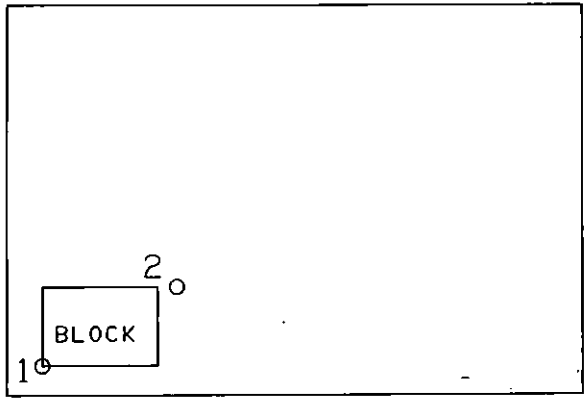
The Block Repeat command makes it possible to duplicate a block a number of times in a straight line.

To use this command, enter BLK REPT and set one point at the location for the first copy, and another point at the location for the second copy of the Block. Then enter the number of times the Block is to be duplicated.

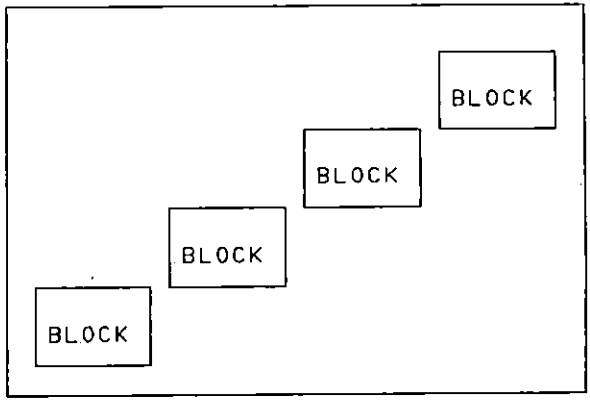
The Block will then be duplicated. The first copy is located with its primary handle on the first point, the second copy is located with its primary handle at the second point, and each subsequent copy is offset the same distance and direction as the first two. See Figure 7.

If the first copy lies directly on the original block, the block will not be duplicated onto itself. Otherwise, the first copy of the Block will be at the first point.

BLOCK REPEAT



BEFORE



AFTER

THE BLOCK REPEAT COMMAND DUPLICATES A BLOCK. TWO POINTS ARE USED - THE LOCATION OF THE FIRST AND SECOND COPIES OF THE BLOCK. THE NUMBER OF COPIES IS THEN ENTERED, AND THE BLOCK IS DUPLICATED THAT MANY TIMES.

FIGURE 7

BLOCK RESET - "Undefine" a Block.

Menu Command: BLK RESET
Short Form: BRE

The Block Reset command resets a Block definition, or "undefines" the Block. This can be used when a new block is to be defined using the Block Add command.

The Block Set command automatically resets any existing block definition. The Block Reset command does not have to be used before the Block Set command.

BLOCK ROTATE - Rotate a Block.

Menu Command: BLK ROTAT
Short Form: BRO

Point 1: Center of rotation (Optional)

The Block Rotate command rotates a block about its primary handle. You will be asked for the number of degrees the block is to be rotated.

The Block is rotated counter-clockwise about the primary handle, or the first point set if a point has been set. (If Geographic angles have been specified in the System Parameter command, then the block will rotate clockwise.)

This command does not make a new copy of the Block - it merely rotates the original block.

BLOCK SAVE - Save a Block to disk.

Menu Command: BLK SAVE
Short Form: BS

Point 1: Primary drawing handle (optional).
Point 2: Secondary drawing handle (optional).

The Block Save command saves a Block to disk as a DesignCAD drawing file. The reference points or handles of the drawing saved to disk may be specified by entering one or two points before the command is entered.

If no points are used with this command, the handles of the block are used for the drawing handles.

BLOCK SET - Define a block.

Menu Command: BLK SET
Short Form: B
Keystroke Command: B

Point 1: Corner of window for entity selection.
Point 2: Opposite corner of window.
Point 3: Primary handle (optional).
Point 4: Secondary handle (optional).

The Block Set command is used to define a Block. 2, 3, or 4 points may be used with this command. The first two points are set in opposite corners of a rectangle, or "window". Every drawing entity entirely within the rectangle is then included in the Block.

The third and fourth points may be used to define the location of the primary and secondary handles of the Block.

If a line or entity lies partly within and partly outside the window, it is not included in the Block. The Cutoff command can be used to break lines around the border of window or section.

The Block Set command overrides any previous Block definition.

BLOCK ZOOM - Zoom a Block.

Menu Command: BLK ZOOM
Short Form: BZ

Point 1: Zoom center (optional).

The Block Zoom command zooms or scales a block larger or smaller. To use this command enter BLK ZOOM. You will be asked for the zoom factor. The block zoom factor is similar to the zoom factor of the Zoom command. Enter 3 for 3 times as large, .5 for half as large, etc.

This command does not make a new copy of the Block - it merely scales the original block. The block is zoomed "into" the primary handle, or into the first point set if any points have been set. In other words, that point of the block will remain in the same place after the zoom operation.

BOX - Draw a box.

Menu Command: BOX
Keystroke Command:]

Point 1: One corner of the box.
Point 2: Opposite corner of the box.

The Box command draws a box or rectangle defined by two points in opposite corners of the box. To use the command, enter BOX and set two points, or set the points and press the "]" key.

The first point is one corner of the box, and the second is the opposite corner. A rectangle will be drawn, with its size and location determined by the two points set.

CHAMFER Chamfer a corner.

Menu Command: CHAMFER
Short Form: CHF
Keystroke Command: ALT-F

Point 1: Set on corner to be chamfered.

The Chamfer command "cuts off" a corner with a straight line. To use the command, enter CHAMFER and set a point at the corner to be rounded, or set the point and press ALT-F. The corner will be cut off with a straight line. The amount of corner removed is determined by the Fillet Radius, as specified in the Drawing Parameter command (DRAWING menu command, or Q keystroke command).

The Fillet command is similar to the Chamfer command, but it "rounds off" the corner with an arc instead of a straight line.

CHANGE - Change the characteristics of a line.

Menu Command: CHANGE
Short Form: CH

Point 1: Set on line or entity to be changed.

The Change command can be used to change the color, layer, line type, or line width of a line or entity. To use this command, enter CHANGE and set a point on the line to be modified.

The following information about the line will be displayed on the screen:

Line Width
Line Type
Line Type Scale
Color
Layer

Any of these can be modified by moving the cursor to the field and entering a new value. After the new values have been entered, press F1. The line will be redrawn with the new values. You can press ESC to return and leave the line in its original form.

The Line Type represents the the line pattern for most drawing entities. It is not used for Hatching entities, and represents the type of format for Dimension entities.

The Line Type Scale is the size of the Line Pattern, or the hatching pattern size for Hatching entities and the text size for Dimension entities.

The Line Width is not used with Hatching, Text, or Dimension entities.

CHANGE COLOR - Change the color of a line.

Keystroke Command: ALT-H

Point 1-n: Set on lines or entities to be changed.

To use this command, set a point on endpoints of all the lines for which you want the color to be changed. Then press ALT-H. You will be asked for the new color, and all the lines or entities selected will be redrawn in the new color.

CHANGE DIRECTORY - Change to a different subdirectory

Menu Command: CHDIR
Short Form: CD

The Change Directory command is used to change to a different subdirectory on the disk drive. To use the command, enter CHDIR. You will then be asked for the new directory name. This command may also be used by entering "CHDIR dirname" from the command line, where "dirname" is the name of the new subdirectory.

This command is similar to the DOS command CHDIR.

The default disk drive can be changed by entering that drive at the command line, for example:

B:

CIRCLE - Draw a circle using center and radius.

Menu Command: CIRCLE
Short Form: CIR
Keystroke Command: O (The letter, not zero).
Keystroke Command: ALT-O

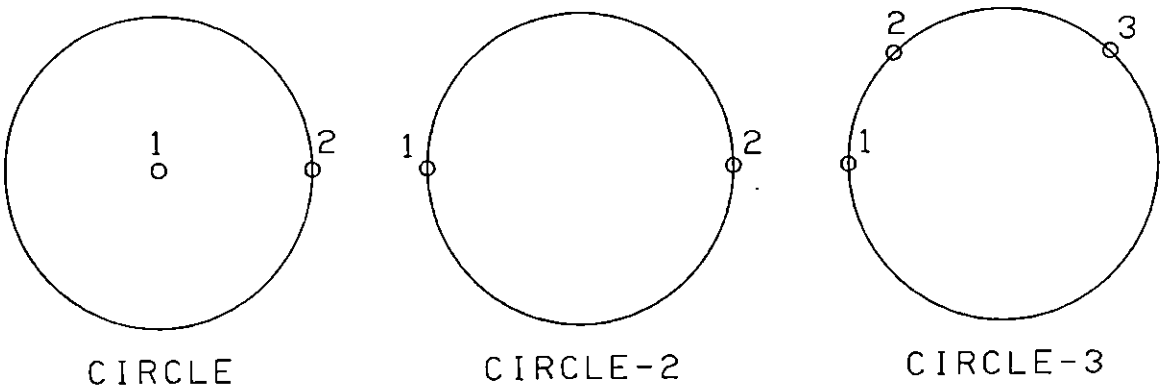
Point 1: Center of the circle.
Point 2: Any point on the circle.

The Circle command is one of three commands used to draw a circle. This command draws a circle based on a point in the center and a point on the outside of the circle.

To use the command, enter CIRCLE and set the two points, or set the points and press the O key (the letter, not the number). A circle will then be drawn through the second point with its center at the first point. See Figure 8.

The ALT-O keystroke command can be used to draw a circle made of short line segments, rather than an actual circle entity. This is sometimes useful if the circle is to be slanted or distorted.

CIRCLE COMMANDS



A CIRCLE CAN BE DRAWN USE THE CENTER AND RADIUS (CIRCLE), THE DIAMETER (CIRCLE-2), OR ANY THREE POINTS ON THE CIRCLE (CIRCLE-3).

FIGURE 8

CIRCLE-2 - Draw a circle using the diameter.

Menu Command: CIRCLE-2
Short Form: C2

Point 1: Endpoint of a diameter of the circle.
Point 2: Other endpoint of the diameter.

The Circle-2 command is one of three commands used to draw a circle. This command draws a circle based on two points set on the diameter of a circle.

To use the command, enter CIRCLE and set two points, one point on each end of the circle's diameter. A circle will then be drawn between the two points. See Figure 8.

CIRCLE-3 - Draw a circle using three points.

Menu Command: CIRCLE-3
Short Form: C3

Point 1: Any point on the circle.
Point 2: Any other point on the circle.
Point 3: Any other point on the circle.

The Circle-3 command is one of three commands used to draw a circle. This command draws a circle through three points.

To use the command, enter CIRCLE, then set the three points through which the circle is to be drawn. There is only one circle that can be drawn through any three points. See Figure 8.

That circle will be drawn through the three points. If the three points lie on a straight line, then a circle cannot be drawn through the points.

This command is very useful when you are using a digitizing tablet to copy a circle.

CLEAR ALL - Clear the screen.

Menu Command: CLEAR ALL
Short Form: CLS
Keystroke Command: Y

The Clear All command erases everything on the screen. To use the command, enter CLEAR ALL or press the Y key. You will be asked if you really want to erase everything. If so, press the Y key. The screen will be cleared.

Be sure you have saved your drawing (if you want to keep it) before you clear the screen - everything on the screen is lost when it is cleared.

When the screen is cleared, the relative zoom, cursor position, and position of the drawing is reset.

COLOR - Change drawing color.

Menu Command: COLOR
Keystroke Command: H

The Color command is used to specify the color to be used in subsequent lines or entities in the drawing. The frame of the drawing area is displayed in the current color.

Your computer may not be capable of displaying the entire range of 16 colors, but it is possible to use the entire range of colors in your drawing. This makes it possible to output your drawing to a plotter that uses more pens than the number of colors your computer can display.

This command changes the color only of lines drawn after the command is executed. The Change Color command (ALT-H Keystroke) or the Change command can be used to change the color of an existing line.

You can determine the color of a line in the drawing with the ID command.

CURSOR TYPE - Change cursor type.

Menu Command: CURS TYPE
Short Form: CT
Keystroke Command: +

The Cursor Type command is used to select the shape of the DesignCAD Cursor. Three cursor types are available:

1. The "x" cursor.
2. The Crosshair.
3. The "+" cursor.

The "x" type cursor is the default cursor that initially appears on the screen. Each time this command is executed, the "next" cursor replaces the current one.

For example, to use the "+" cursor, enter CURS TYP or press the "+" key twice. To go back to the "x" cursor, enter CURS TYP or press "+" another time.

CURVE - Draw a curve.

Menu Command: CURVE
Short Form: C
Keystroke Command: C

Points 1-n: Points through which the curve is drawn.

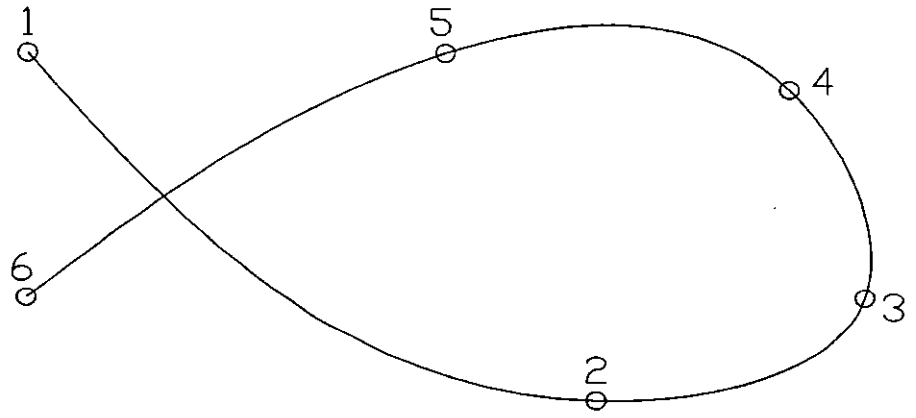
The Curve command draws a curve through three or more points. To use the Curve command, enter CURVE and set the points on the screen through which a curve is to be drawn, or set the points and press the C key. A curve will be drawn through each point, beginning with the first and continuing to the last point.

Figure 9 shows a curve drawn through several points. The points shown are enlarged for illustration. They do not normally appear after the line is drawn.

The mathematical formula used to determine the curve is a cubic spline. This command can also be performed by pressing the fourth button on a mouse or digitizing tablet.

The ALT-C keystroke command can be used to draw a curve made of short line segments, rather than an actual Curve entity. This is sometimes useful if the curve is to be slanted or distorted, and can be done to facilitate editing of the curve.

THE CURVE COMMAND



THE CURVE COMMAND DRAWS A CURVE THROUGH A SET OF POINTS.

FIGURE 9

CUTOFF - Break lines around a section.

Menu Command: CUTOFF
Short Form: CUT

Point 1: Corner of the window for entity selection.
Point 2: Opposite corner of the window.

The Cutoff command is used to break lines around a section or "window". To use the command, enter CUTOFF and set two points in opposite corners of a rectangle.

Each line that crosses the edge of the rectangle will be broken at the edge of the rectangle. The lines will not look any different, but each broken line will be made up of two separate entities afterward.

Text, attributes, hatching patterns, and dimension information are not affected by this command. Lines, curves, circles, arcs, ellipses, and elliptical arcs are broken.

This command can be used to just break a line, or can be used in conjunction with the Set command to "cut off" a section for a Block definition. It can also be used before a Section command to move, copy, delete, or save partial lines.

DIMENSION - Dimension horizontally or vertically.

Menu Command: DIMENSION
Short Form: DIM

Point 1: First point to be measured.
Point 2: Second point to be measured.
Point 3: Location for dimension information(optional)

The Dimension command is used to draw dimensioning information automatically. Three points are used with this command. The first two points are the points to be measured. The horizontal or vertical distance is measured between the first two points. The third point is the location for the dimensioning information.

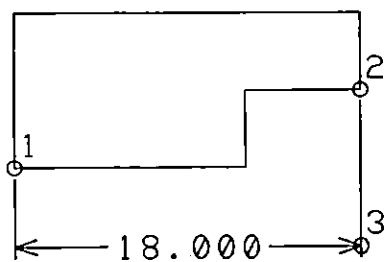
If the third point is below or above the other two points, the dimension will be done horizontally. Otherwise, it will be done vertically. See Figure 10.

The dimension information can be maintained dynamically by enabling "Dynamic Dimensioning" in the System Parameter command.

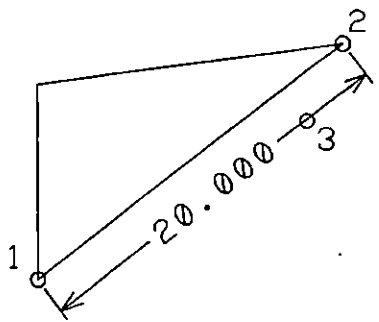
When Dynamic Dimensioning is enabled, the distance displayed will change automatically if the drawing units are changed. Also, the distance displayed will change if the points of the Dimension entity are moved with the Section Stretch or Point Move command.

When Dynamic Dimensioning is disabled, the dimension information takes the form of text and lines. The dimensions can then be edited, and will not change if the drawing units change or if the dimension points are moved.

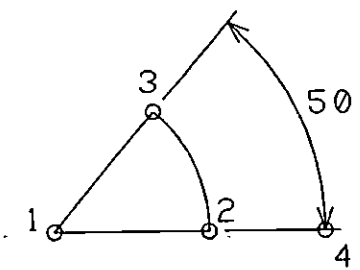
AUTO DIMENSIONING



DIMENSION



DIMENSION LINE



DIMENSION ANGLE

THERE ARE THREE COMMANDS FOR AUTO DIMENSIONING.

FIGURE 10

DIMENSION ANGLE - Dimension an angle.

Menu Command: DIM ANGLE
Short Form: DA

Point 1: Center point of the angle to be measured.
Point 2: Beginning point of the angle.
Point 3: Ending Point of the angle.
Point 4: Leader line start (optional)
Point 5: Leader line end (optional)
Point 6: Leader line start (optional)
Point 7: Leader line end (optional)

This command is used to draw dimensioning information automatically. Three to seven points are used with this command. The first point is the center of the angle to be measured. The second and third points represent the beginning and ending of the angle to be measured.

The fourth and fifth points may optionally be used to specify the beginning and ending points for the leader lines. The leader line will be drawn from the fourth to the fifth point, as shown in Figure 10. A sixth and seventh point can be used to specify the location of the second leader line if it is different from the first.

The dimension arc is drawn from the second point to the angle of the third point.

The dimension information can be maintained dynamically by enabling "Dynamic Dimensioning" in the System Parameter command.

When Dynamic Dimensioning is enabled, the distance displayed will change automatically if the if the points of the Dimension entity are moved with the Section Stretch or Point Move command.

When Dynamic Dimensioning is disabled, the dimension information takes the form of text, lines, and arcs. The dimensions can then be edited, and will not change if the dimension points are moved.

DIMENSION LINE - Dimension at any angle.

Menu Command: DIM LINE
Short Form: DL

Point 1: First point to be measured.
Point 2: Second point to be measured.
Point 3: Location for dimension information(optional)

This command is used to draw dimensioning information automatically. Three points are used with this command. The first two points are the points to be measured, and the third point is the location for the dimensioning information.

The distance measured is the "straight line" distance between the first two points, at the angle between those points. See Figure 10.

The dimension information can be maintained dynamically by enabling "Dynamic Dimensioning" in the System Parameter command.

When Dynamic Dimensioning is enabled, the distance displayed will change automatically if the drawing units are changed. Also, the distance displayed will change if the points of the Dimension entity are moved with the Section Stretch or Point Move command.

When Dynamic Dimensioning is disabled, the dimension information takes the form of text and lines. The dimensions can then be edited, and will not change if the drawing units change or if the dimension points are moved.

DIRECTORY - Display DesignCAD drawings on disk.

Menu Command: DIRECTORY
Short Form: DIR
Keystroke Command: F7

The Directory command allows you to display the names of drawings that have been stored on disk, and to delete or erase any of the drawings from the disk. To use the command, enter DIRECTORY or press the F7 key.

First, you will be asked whether you want to display 2-Dimensional (ordinary drawings), 3-Dimensional (from DesignCAD 3-D), or ProDesign II files.

2-Dimensional drawings have a file extension of .DC2, 3-Dimensional drawings have a file extension of .DC3, and ProDesign II drawings have a file extension of .PD1.

Next, you will be asked for the drive to be searched. If you want to search the default drive (the drive on which you are currently operating), press ENTER. To display the drawings from another drive, enter the drive name, such as B: .

The names of all the drawings will be displayed on the screen. After the drawings are displayed on the screen, you will be given the option to delete any of them.

If you are using a hard drive, the default directory will be searched for drawings. The Change Directory command can be used to change to another directory.

DRAWING PARAMETER - Enter drawing parameters.

Menu Command: DRAWING
Short Form: DP
Keystroke Command: Q

The Drawing Parameter command is used to specify DesignCAD drawing parameters, such as Cursor Step Size, Default Text Size and Angle, and Grid information.

To use the command, enter DRAWING from the PARAMETER submenu, or press the Q key. A screen as shown in Figure 11 will be displayed. To change one of the parameters, move the cursor to that location and enter the new value. The drawing parameters are described below.

Many of the drawing parameters use Drawing Units as units of distance. A drawing unit is the distance specified by the Units command. It is the unit of distance used throughout the drawing.

1. COLOR
The Color parameter defines the color in which subsequent lines are to be drawn. The drawing screen frame is displayed in the current color. The default color is color 1, or white.

2. CURSOR STEP SIZE
The Cursor Step Size determines the distance the cursor travels when one of the numeric keypad keys is pressed. A cursor step size of one causes the cursor to be moved a distance of one drawing unit when a cursor key is pressed. The initial cursor step size is 1.

PRODESIGN II DRAWING PARAMETERS

COLOR (1-16):	1
CURSOR STEP SIZE:	1.000
DEFAULT TEXT SIZE:	1.00
DEFAULT TEXT ANGLE:	0.0
LINE TYPE (0-8):	0
LINE TYPE SCALE (1 = NORMAL):	1.00
LINE WIDTH (0 FOR A SINGLE LINE):	0.00
DISPLAY GRID TYPE (1-3):	0
DISPLAY GRID SIZE:	4.000
SNAP GRID SIZE:	0.000
FILLET RADIUS:	0.500

USE ARROW KEYS TO SELECT AN ITEM TO BE CHANGED.
PRESS "F1" TO USE NEW VALUES (BUT NOT SAVE THEM).
PRESS "F2" TO USE NEW VALUES AND SAVE THEM AS DEFAULTS.
PRESS "ESC" TO ABORT.

FIGURE 11

When the drawing is zoomed to become larger or smaller, the Cursor Step Size and the Display Grid remain the same size relative to the screen, not to the drawing. When you return to the Drawing Parameter command after a zoom, the Cursor Step Size and Display Grid may change in value, but they are the same size on the screen.
Your display may have vertical dots spaced differently than the horizontal dots on the screen. If this is the case, the vertical cursor steps may not always appear to be the same distance as the horizontal steps. This will not affect the final appearance of your drawing.

3. DEFAULT TEXT SIZE
The default text size determines the size at which text is to be displayed when only one point (or two points in the same location) is used with the Text command. The Text size is the height of the text in drawing units.

If two or more points (at different locations) are used with the Text command, the text size and angle are determined by the positions of those points.

The default text size also determines the size of dimensioning labels and arrowheads.

4. DEFAULT TEXT ANGLE
The default text angle determines the angle at which text is to be displayed when only one point (or two points in the same location) is used with the Text command.

With mathematical angles, an angle of 0 is normal horizontal text, 90 is vertical beginning at the bottom, 180 is upside down, etc. If geographical angles have been selected in the System Parameter command, then 0 will be vertical, 90 normal horizontal text, etc.

If two or more points (at different locations) are used with the Text command, the text size and angle are determined by the positions of those points.

5. LINE TYPE
Line Type is used to select different dot or dash patterns for lines in the drawing. There are 8 patterns available, as shown in Figure 12. 0 is used for solid lines.

The line type you choose will be used until it is changed again. To display the available line types, enter "?" for the Line Type.

6. LINE TYPE SCALE
The line type scale represents the relative pattern size for the line - in other words, it tells how far apart the dashes are. A value of 1 is normal.

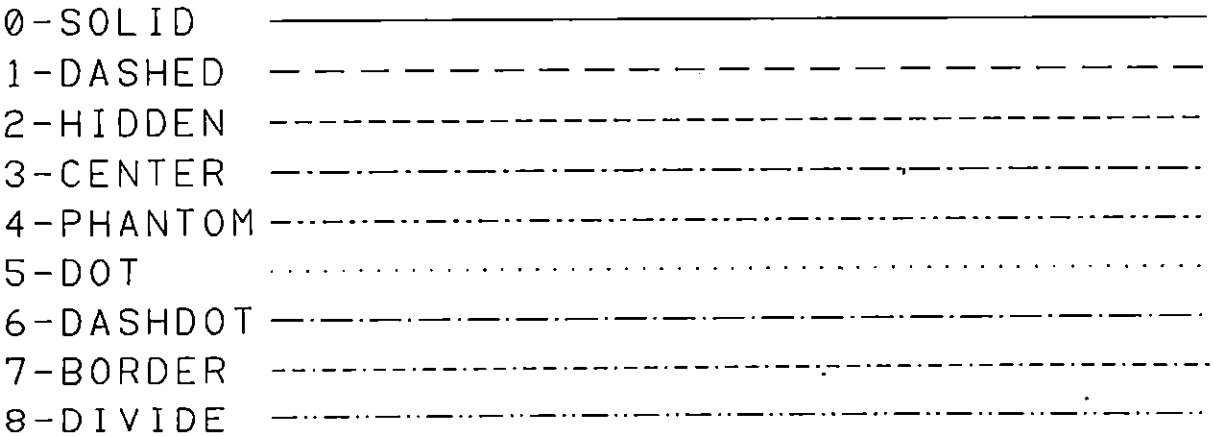
7. LINE WIDTH
The Line Thickness is the width or thickness of the lines used in the drawing. Enter 0 for single line thicknesses, 1 for 1 drawing unit wide, .2 for .2 drawing units wide, etc.

8. DISPLAY GRID TYPE
DesignCAD can display a grid on the screen as a visual aid. There are three different display grid types:

1 - Solid Lines
2 - Border Lines
3 - Dots

After the grid type is selected, it can be enabled using the GRID command.

LINE TYPES



ANY OF THESE LINE TYPES CAN BE USED IN CURVED AS WELL AS IN STRAIGHT LINES. THE LINE TYPE SCALE DETERMINES THE LENGTH OF THE DASHES.

FIGURE 12

9. DISPLAY GRID SIZE
The display grid size is the distance between grid lines or dots, in Drawing Units. The default display grid size is 4 Drawing Units.

When the drawing is zoomed to become larger or smaller, the Cursor Step Size and the Display Grid remain the same size relative to the screen, not to the drawing. When you return to the Drawing Parameter command after a zoom, the Cursor Step Size and Display Grid may change in value, but they are the same size on the screen.

10. SNAP GRID SIZE
The Snap Grid is used to define the resolution at which points can be set. The snap grid is an imaginary grid placed over your drawing. Each point you set with the 0 key (or the first mouse or digitizer button) will be located exactly on one of the grid points.

The distance between the points on the snap grid is the Snap Grid Size. The units of measurement used for the snap grid size are the Drawing Units defined by the Units command.

Suppose, for example, you are making a drawing on which every point should lie on 1/4 inch spacing. First, use the Units command to define an inch on the screen. Then, use the Snap Grid command and enter a snap grid size of .25 . Subsequent points you set on the drawing will be set at the nearest 1/4" point.

The Snap grid can be enabled or disabled with the Snap Grid command (SNAP). The Snap Grid does not affect points set with the Gravity Point, Point XY, Point Relative, or Point Polar commands.

11. FILLET RADIUS
The fillet radius is the radius used for the Fillet or Chamfer command (F or ALT-F keys). The radius is entered in Drawing Units.

If you change the size of the drawing with the Zoom command, those drawing parameters that use Drawing Units will cover the same number of Drawing Units, but will be larger (or smaller) on the screen.

If you change the drawing units of measurement with the Units command, the drawing parameters will cover the same area on the screen but will have a different numeric value, according to the change in Drawing Units.

The Drawing Parameter command can be terminated three ways:

F1 - Use the system parameters but do not save them to disk.
F2 - Save the parameters as program defaults. (Use the system parameters every time the program begins.)
ESC - Return and don't change anything.

Drawing Units, in addition to several of the the Drawing and System Parameters, are saved when the Drawing Parameter command is terminated with the F2 key. The parameters saved as defaults are:

1. Cursor increment.
2. Display Grid information.
3. Snap Grid information.
4. Precision.
5. Character Font.
6. Default Text Size and Angle.
7. Drawing Units per inch on output.
8. Fillet Radius.
9. Attribute Display on or off.
10. Drawing Units size.
11. Menu on or off.
12. Mathematical or Geographical angles.
13. Fill Wide Lines on or off.

14. Sound Level.
15. Edit current section only - on or off.
16. Drawing origin.
17. Save parameters with drawing.
18. Dynamic Dimensioning enabled.
19. Dimension format.

ELLIPSE - Draw an ellipse or Oval.

Menu Command: ELLIPSE
Short Form: O
Keystroke Command: O

Point 1: Center point of the ellipse.
Point 2: One axis of the ellipse.
Point 3: The other axis of the ellipse (optional).

The Ellipse command is used to draw a circle or ellipse. To use the command, enter ELLIPSE, or set points and then press the O key. Set a point at the center of the desired ellipse. Set another point at the major or minor axis of the ellipse itself. Set a third point anywhere on the ellipse itself.

DesignCAD will draw an ellipse that passes through the second and third points with its center at the first point. The relative horizontal radius (the major or minor radius) is located between the first two points.

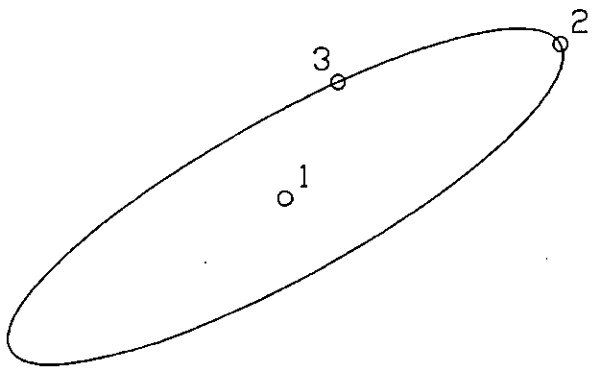
The relative height of the ellipse depends on the position of the third point - DesignCAD will draw the ellipse so that it passes through the third point. If the third point is out of range (if an ellipse cannot be drawn through it), then a circle will be drawn.

The term horizontal radius is used here in a relative manner. The major (or minor) radius of the ellipse is defined to be the imaginary line drawn from the first point to the second - even if it is not actually horizontal. It is possible to draw an ellipse at an angle by specifying the first two points at that angle.

Figure 13 gives an example of this command.

The ALT-O keystroke command can be used to draw an ellipse made of short line segments, rather than an actual circle or ellipse entity. This is sometimes useful if the ellipse is to be slanted or distorted.

THE ELLIPSE COMMAND



TO DRAW AN ELLIPSE, SET THE FIRST POINT IN THE CENTER, THE SECOND POINT ON THE MAJOR OR MINOR AXIS, AND THE THIRD POINT ANYWHERE ELSE ON THE ELLIPSE.

FIGURE 13

ELLIPTICAL ARC - Draw an elliptical arc.

Menu Command: ELLIP ARC
Short Form: EA
Keystroke Command: A

Point 1: Center point of the ellipse.
Point 2: Arc Beginning.
Point 3: Arc End.
Point 4: One of the axes of the ellipse (optional).

The Elliptical Arc command is used to draw an elliptical arc (a portion of an ellipse or oval) between two points. To use the command, set 3 or 4 points:

1. The center of the circle or ellipse.
2. The beginning point that lies on the circle or ellipse.
3. The ending point that lies on the circle or ellipse.
4. (Optional) The point representing the Major (or Minor) radius of the arc.

A line is drawn beginning at the second point, clockwise around the circle or ellipse to the third point. The position of the third point determines whether the arc will follow a circular or elliptical path. See Figure 14.

The fourth point can be used to specify the major (or minor) axis for an elliptical arc, if it does not lie on the first point. If the fourth point is used, the arc will begin at the angle of the second point, but may not pass precisely through that point.

If the second and third points are the same distance from the first, then a circular arc will be drawn.

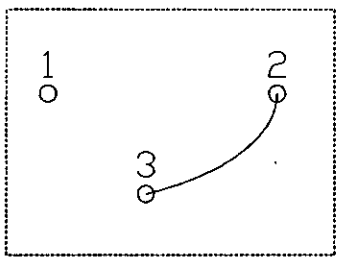
The ALT-A keystroke command can be used to draw an arc made of short line segments, rather than an

actual Arc entity. This is sometimes useful if the arc is to be slanted or distorted.

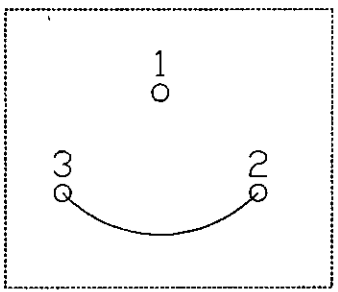
If the keystroke command A or ALT-A is used with only two points set, an circular arc will be drawn. See the Arc command.

THE ELLIPTICAL ARC

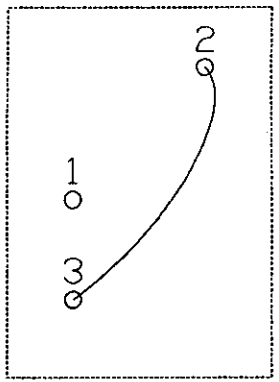
EXAMPLE 1



EXAMPLE 2



EXAMPLE 3



THE FIRST POINT IS THE CENTER, THE SECOND IS THE RADIUS, AND THE THIRD POINT CONTROLS THE ELONGATION.

FIGURE 14

END - Stop the program

Menu Command: END
Keystroke Command: F8

This command terminates the DesignCAD session and returns to the operating system. Be sure you have saved anything you want to keep before stopping the program with this command.

ERASE - Erase a line or entity.

Menu Command: ERASE
Keystroke Command: E

Point 1: Set on line or entity to be erased.

The Erase command is very helpful in making corrections to drawings. This command allows you to delete any of the lines, curves, circles, etc. from your drawing.

To erase a line with this command, set a point on the line to be erased, or on one of the points that was used to create the line (such as one of the end points or one of the corners). The line or drawing entity will then be deleted from the drawing.

This command can be used to erase any drawing entity: Lines, Curves, Arcs, Circles, Ellipses, and Text.

The Oops command can be used to replace items accidentally deleted with the Erase command.

FILLET - Fillet a corner.

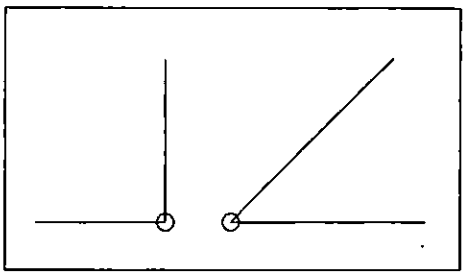
Menu Command: FILLET
Short Form: F
Keystroke Command: F

Point 1: Set on corner to be filleted.

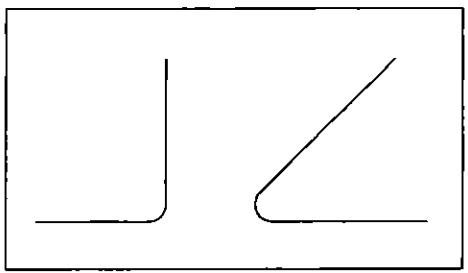
The Fillet command "rounds off" a corner with a smooth arc. To use the command, enter FILLET and set a point at the corner to be rounded, or set the point and press the F key. The corner will be rounded with an arc. The size of the arc is determined by the Fillet Radius, as specified in the Drawing Parameter command. See Figure 15.

The Chamfer command is similar to the Fillet command, but it "rounds off" the corner with a straight line.

THE FILLET COMMAND



BEFORE



AFTER

THE FILLET COMMAND IS USED TO ROUND OFF CORNERS. THE FILLET RADIUS IS SET USING THE "DRAWING PARAMETER" COMMAND.

FIGURE 15

GRAVITY MOVE - Move the cursor to the nearest point.

Menu Command: GRAV MOV
Short Form: GM
Keystroke Command: ,

The Gravity Move command is used to move the cursor to the nearest point in the drawing (or the nearest point that has been set). To use the command, enter GRAV MOV or press the "," key. The cursor will move to the nearest existing point in the drawing.

If the current drawing has been zoomed in and out, or if it has been retrieved at various sizes, the points of the drawing may not lie exactly on the dots or pixels on the screen. This command allows you to access these points to manipulate these lines with accuracy.

GRAVITY POINT - Move the cursor to the nearest point in the drawing and set a point there.

Menu Command: GRAVITY
Short Form: GV
Keystroke Command: .

The Gravity Point command is used to set a point exactly on top of another point that has been set or previously used in the drawing. To use the command, enter GRAVITY or press the "." key. The cursor will move to the nearest existing point and set a point at that location.

If the current drawing has been zoomed in and out, or if it has been retrieved at various sizes, the points of the drawing may not lie exactly on the dots or pixels on the screen. This command allows you to access these points to manipulate these lines with accuracy.

For example, you may want to set a point exactly at the end-point of a line in the drawing. Move the cursor near the end-point, and enter GRAVITY or press the "." key. The cursor will move to that location and a point will be set there. The point set will be set exactly on the end-point of the line, even if that location does not correspond to a dot on the screen.

GRID - Turn the display grid on or off.

Menu Command: GRID
Short Form: G
Keystroke Command: G

The Grid command enables or disables the Display Grid, and causes it to be drawn on the screen. The type of grid displayed is determined by the Drawing Parameter command. To turn the display grid on or off, enter GRID or press the G key.

HATCH - Fill an area with a pattern.

Menu Command: HATCH
Short Form: HA
Keystroke Command: #

Point 1-n: Points outlining area to be hatched.

The Hatch command is used to fill an area with a pattern. Any of several patterns may be used. These patterns are displayed in Appendix G.

To use the menu command, enter HATCH and set points around the area to be filled with the pattern, or hatched. Then press ENTER. To use the keystroke command, set the points and then press #. See Figure 16.

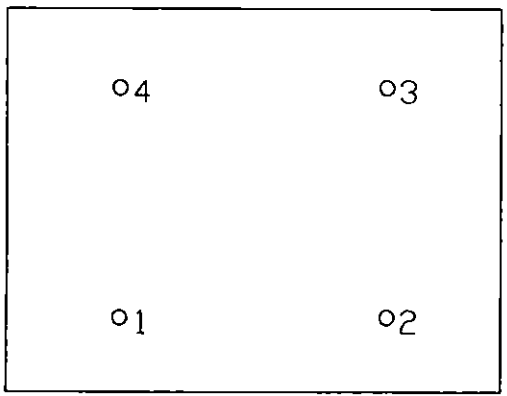
You will be asked for the hatching pattern. Enter the name or number of the pattern to be used.

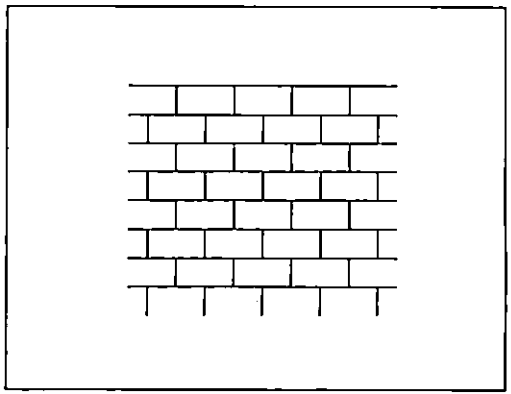
You will then be asked for the scale and angle of the pattern. The scale is the size of the pattern elements, and the angle is the angle at which the pattern is to be drawn. The pattern will then be drawn inside the polygon defined by the points that were set.

The hatching pattern is a single entity - it is not possible to erase, snap to, or manipulate one of the lines in the pattern. To erase a hatch pattern, set a point on one of the points used to define the border of the hatching and enter ERASE (or press the E key).

Custom hatch patterns can be created and used with DesignCAD. For details on this, see Appendix J.

THE HATCH COMMAND





TO USE THE HATCH COMMAND, SET POINTS AROUND THE AREA TO BE FILLED.

FIGURE. 16

HATCH LINE - Fill an existing line with a pattern.

Menu Command: HATCHL
Short Form: HL

Point 1: Set point on line to be hatched.

The Hatchl command is used to fill an area with a pattern. This command is similar to the hatch command, except the area to be hatched is the area within an existing line in the drawing instead of the area within a set of points.

To use the command, enter HATCHL. Then set a point on the line to hatched. You will be asked for the hatching pattern. Enter the name or number of the pattern to be used.

You will then be asked for the scale and angle of the pattern. The scale is the size of the pattern elements, and the angle is the angle at which the pattern is to be drawn.

The pattern will then be drawn inside the polygon defined by the line specified. If the line is not in the form of a polygon, an imaginary line will be "drawn" from the last to the first point of the line. This command hatches only the area inside a single line entity.

To hatch inside a circle, curve, etc., the circle or curve must be drawn as a a set of vectors. This can be done using the ALT-O or ALT-C keystroke command.

The hatching pattern is a single entity - it is not possible to erase, snap to, or manipulate one of the lines in the pattern. To erase a hatch pattern, set a point on one of the points on the border of the hatching and enter ERASE (or press the E key).

Custom hatch patterns can be created and used with DesignCAD. For details on this, see Appendix J.

HELP - Display menu commands.

Menu Command: HELP
Keystroke Command: ?

The Help command displays the names of all the DesignCAD menu commands on the screen. To use the Help command, enter HELP or press the ? key. Then press ENTER to go from one page to the next. If you want to return to the command mode before you have finished Help, press the ESC key.

ID - Display information about a line.

Menu Command: ID
Keystroke Command: I

Point 1: Set point on line or entity to be ID'd.

The ID command displays the color, linetype, and layer of a line or drawing entity. It also displays the area and length of the entity.

To use the menu command, enter ID and set a point on the line or entity you want information on. to use the keystroke command, set the point and then press the I key. The information about that line will be displayed at the bottom of the screen.

This command has two primary functions. First, to determine the characteristics of a line or entity.

Second, this command can be used to determine the length and area of irregular objects. For example, you can display the length of a curve, the circumference of an ellipse, or even the area inside a curved line. The only restriction is that the area and length can be displayed only for a single entity.

INTERSECT - Snap to the nearest intersection.

Menu Command: INTERSECT
Short Form: INT

Point 1: Set point on first intersecting line.
Point 2: Set point on second intersecting line.
(Points set only after command entry.)

The Intersect command is used to set a point at the intersection of two lines. The two lines may be lines, circles, arcs, ellipses, or elliptical arcs. This command does not find the intersection of two ellipses or of a curve and another line.

To use the Intersect command, enter INTERSECT and set a point on each of the two intersecting lines. The cursor will then move to the intersection and a point will be set there.

This command temporarily erases any previously set points, and replaces them after the command is complete. For example, you might have three points set and want to set a fourth point at an intersection. After INTERSECT is entered, the original 3 points are erased. You then set two points to designate the intersecting lines. The two points just set are erased, the original 3 points are brought back, and the fourth point is set at the intersection of the line.

This is the only menu command that does not allow the points to be set before the command is entered.

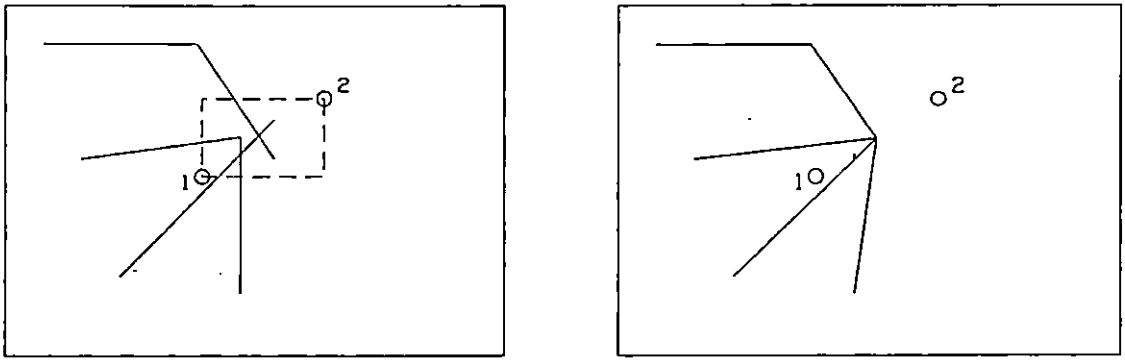
JOIN - Join two or more endpoints.

Menu Command: JOIN

Point 1: Corner of window of points to be joined.
Point 2: Opposite corner of window.

The Join command is used to connect the points of two or more lines in the drawing. To use the command, set two points in opposite corners of a rectangle containing the endpoints to be joined. The enter JOIN. All the points of lines inside the rectangle will be moved to the "geographic center" of the points, connecting them. See Figure 17.

JOIN COMMAND



THE JOIN COMMAND IS USED TO JOIN TWO OR MORE POINTS TO THE SAME LOCATION. SET TWO POINTS IS OPPOSITE CORNERS OF A WINDOW THAT CONTAINS ALL THE POINTS TO BE JOINED.

FIGURE 17

LAYER - Layer command.

Menu Command: LAYER
Short Form: LAY
Keystroke Command: L

A DesignCAD drawing can have up to twenty one layers or subdivisions. Dividing a drawing into layers can be useful because you have the ability to plot or print only certain layers, and because the Layer command allows a layer to be saved, deleted, moved from one layer to another, or changed in color or line type. (It is possible to have more than one color and line type in a layer.)

When you create a drawing, the commands are added to the Current Layer. The Current Layer is layer 1, unless otherwise specified.

Each layer can be set to active or inactive status. An active layer is displayed on the screen, and an inactive layer is "invisible".

To set the Current Layer, to set the active status of the layers, or to name layers, enter LAYER or press the L key. You will see a screen like the one shown in Figure 18.

You can set the current and active layers, and can give a name to any layer. After entering the information, press F1 to continue with the drawing.

Only one layer is the Current Layer at a time. All lines drawn are added to the current layer.

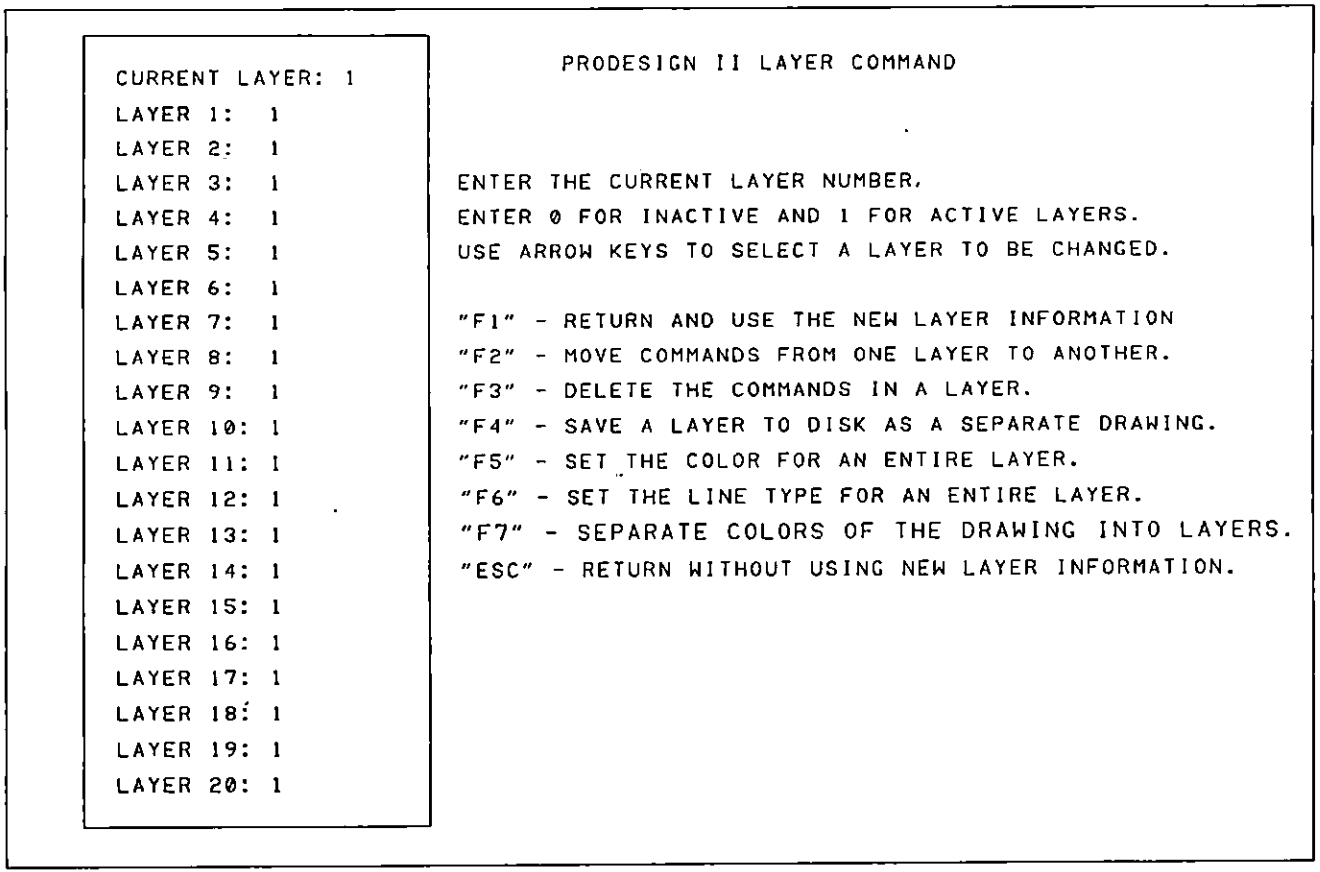

CURRENT LAYER: 1
LAYER 1: 1
LAYER 2: 1
LAYER 3: 1
LAYER 4: 1
LAYER 5: 1
LAYER 6: 1
LAYER 7: 1
LAYER 8: 1
LAYER 9: 1
LAYER 10: 1
LAYER 11: 1
LAYER 12: 1
LAYER 13: 1
LAYER 14: 1
LAYER 15: 1
LAYER 16: 1
LAYER 17: 1
LAYER 18: 1
LAYER 19: 1
LAYER 20: 1
PRODESIGN II LAYER COMMAND
ENTER THE CURRENT LAYER NUMBER,
ENTER 0 FOR INACTIVE AND 1 FOR ACTIVE LAYERS.
USE ARROW KEYS TO SELECT A LAYER TO BE CHANGED.
"F1" - RETURN AND USE THE NEW LAYER INFORMATION
"F2" - MOVE COMMANDS FROM ONE LAYER TO ANOTHER.
"F3" - DELETE THE COMMANDS IN A LAYER.
"F4" - SAVE A LAYER TO DISK AS A SEPARATE DRAWING.
"F5" - SET THE COLOR FOR AN ENTIRE LAYER.
"F6" - SET THE LINE TYPE FOR AN ENTIRE LAYER.
"F7" - SEPARATE COLORS OF THE DRAWING INTO LAYERS.
"ESC" - RETURN WITHOUT USING NEW LAYER INFORMATION.

FIGURE 18

There are several DesignCAD commands that modify or erase existing lines or entities. These commands can operate on the current layer only, or an option can be selected to allow lines or entities in all visible layers to be affected.

This option is set using the System Parameter command. You should be aware of this option's setting, because it influences the effect of a number of DesignCAD commands. See System Parameter command. With the Layer command, you can also save a layer, delete all the commands in a layer, move commands from one layer to another, or change the color or line type of all the commands in a layer. To perform any of these functions, press the appropriate function key under the Layer command.

Another option with the Layer command is separating the drawing into layers based on color. In other words, everything in color 1 can be moved to layer 1, color 2 moved to layer 2, etc. This alters the layering information of the entire drawing.

A layer is different from an overlay. An overlay is a separate drawing retrieved with the Overlay command for viewing purposes only. A layer is a subdivision of the current drawing that can be manipulated with the Layer command.

When a drawing is retrieved, commands in layer 0 are retrieved into the Current Layer, even if the Current Layer is not 0. Other layers are retrieved into their respective layers. This allows symbols to be saved in layer 0 and retrieved into the Current Layer.

LINE - Draw a line.

Menu Command: LINE
Short Form: V
Keystroke Command: V

Point 1-n: Points the line is to be drawn through.

The Line command is used to draw straight lines between points. To use this command, enter LINE and set the points to be connected on the screen. After the points have been set, press ENTER. To use the keystroke command V, set the points and then press the V key.

A line or vector will be drawn through the points, beginning at the first point and continuing to the last point. Figure 19 shows a vector drawn through four points.

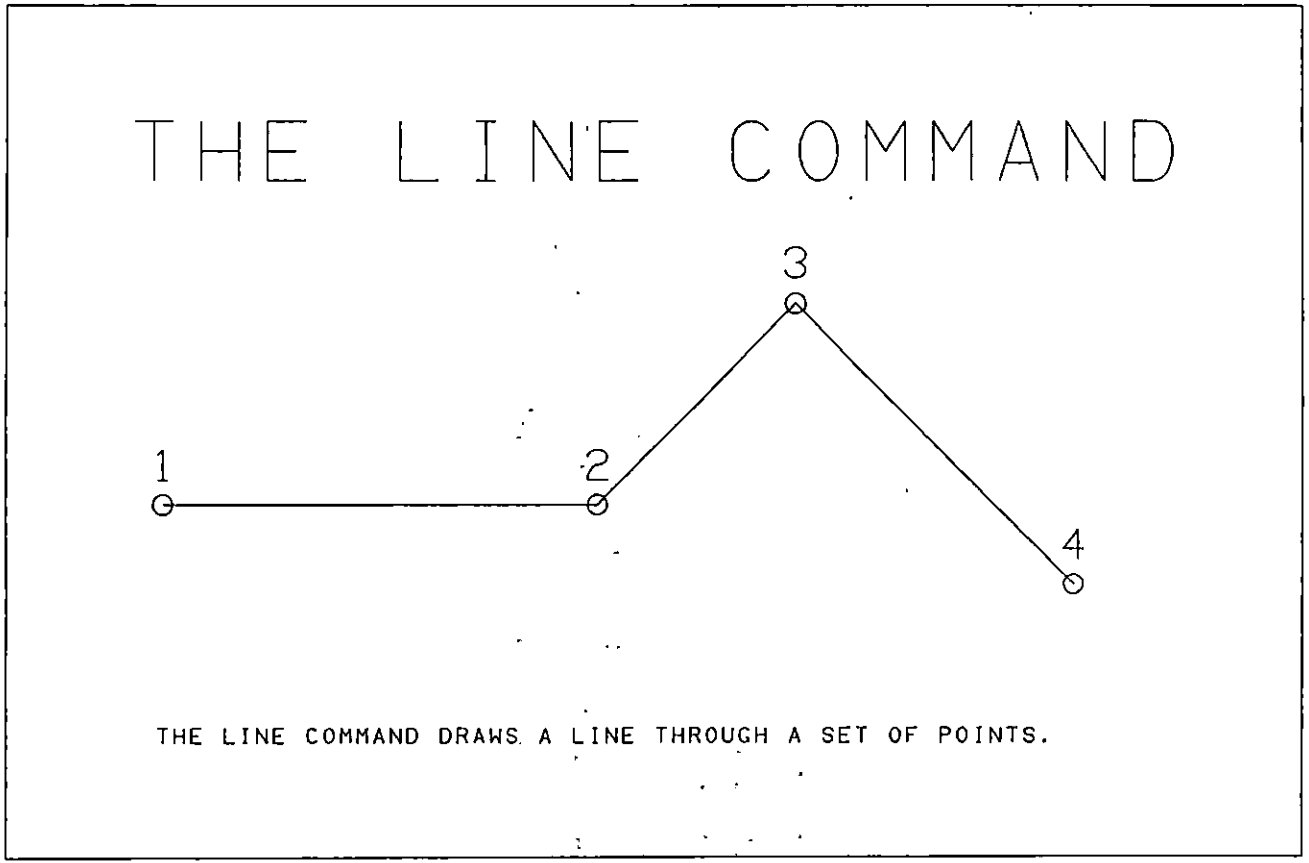

THE LINE COMMAND
1
2
3
4
THE LINE COMMAND DRAWS A LINE THROUGH A SET OF POINTS.

FIGURE 19

LINE TYPE - Select the line type.

Menu Command: LINE TYPE
Short Form: LT
Keystroke Command: /

The Line Type command can be used to select different dot or dash patterns for lines in the drawing. There are 8 patterns available, as shown in Drawing Parameter command.

To use this command, enter LINE TYPE or press the "/" key. You will be asked which line type is to be used. Make your selection, 0 through 8 (0 is a solid line). The line type you choose will be used until it is changed again with this command. To display the available line types, enter "?" for the line type.

After you select the appropriate line type, you will be asked for the line type scale. The line type scale represents the relative pattern size for the line - in other words, it tells how far apart the dashes are. A value of 1 is normal.

LINESNAP - Snap to the nearest line.

Menu Command: LINESNAP
Short Form: LS
Keystroke Command: K

The Linesnap command moves the cursor to the nearest line or entity and sets a point there. To use the command, enter LINESNAP or press the K key. The cursor will then move to the nearest location on the nearest line, and a point will be set there.

MACRO DEFINE - Define a Macro.

Menu Command: MAC DEF
Short Form: MD

A Macro is a set of keystrokes that can be saved and later used as a single command. This is a very powerful DesignCAD feature.

The Macro Define command is used to create a Macro. To create a Macro, enter MAC DEF. You will then be asked for the Macro name. A Macro name can be 1 to 8 characters, and may contain letters and numbers only.

After you have entered the Macro name, you will return to the drawing screen. Every keystroke entered will become part of the Macro. If your display has 640 or better horizontal resolution, the word MACRO will appear at the top of the screen while the Macro is being created.

The macro consists only of keystrokes. Any movement of the cursor from a mouse or digitizing tablet will not be saved with the macro.

After you have finished building the Macro, use the Macro Save command (& key) to save it. The Macro can then be executed using the MAC EXEC command or by entering &MACNAME, where MACNAME is the name of the Macro.

Suppose, for example, you wanted to make a Macro to draw a box 1 cursor step wide and one cursor step tall. The macro will be named BOXMAC. To do this, enter the command MAC DEF, and enter the name of the Macro when requested: BOXMAC.

Then, on the drawing screen, draw the box. Now, enter the MAC SAVE command. The Macro will be saved

to disk. To execute the Macro and draw a box at the current cursor position, use the MAC EXEC command or enter &BOXMAC.

There are 10 special names that can be used with Macros so that the macros can be executed by pressing a single key. Macros MACRO1 through MACRO10 can be executed by pressing ALT-F1 through ALT-F10, respectively. For example, if you had named the box Macro MACRO3 instead of BOXMAC, you could use it to draw a box at the current cursor position by pressing ALT-F3.

The macro execution begins at the last point set. If no points have been set, then the macro execution begins at the current cursor position.

MACRO EXECUTE - Execute a Macro.

Menu Command: MAC EXEC
Short Form: EXEC

The Macro Exec command is used to execute a Macro. To use the command, enter MAC EXEC, and then enter the Macro name when requested. That Macro will be performed at the current cursor position.

It is also possible to execute a Macro by entering:

&MACNAME

where MACNAME is the name of the Macro. Macros named MACRO1 through MACRO10 can be executed by pressing ALT-F1 through ALT-F10, respectively.

The macro execution begins at the last point set. If no points have been set, then the macro execution begins at the current cursor position.

MIDPOINT - Snap to the midpoint of a line.

Menu Command: MIDPOINT
Short Form: MID

The Midpoint command moves the cursor to the center of the nearest straight line and sets a point there. To use the command, enter MIDPOINT. The cursor will then move to the nearest location on the nearest line, and a point will be set there.

OOPS - "Unerase" the last thing erased.

Menu Command: OOPS
Keystroke Command: !

The Oops command is used to reverse an undesired Undo, Erase, or Section Delete command. To use the command, enter OOPS or press the "!" key. If the last command was an Undo, Erase, or Section Delete command, then the last line or entity that was erased will be redrawn on the screen. It is possible to reverse several erasures with the Oops command, provided the erasures were done since any other drawing command.

ORIGIN - Move the origin of the drawing.

Menu Command: ORIGIN

Point 1: Location of the new origin.

This command is used to move the origin to another location. The origin is the location of coordinate 0,0. It lies in the lower left corner of the screen before any zooms, pans, rotates, etc. are performed. To use the command, set a point at the location for the new origin and enter ORIGIN. Until it is changed again, the origin will be at that point.

Zoom, Rotate, and Pan may change the location of the origin on the screen, but it remains at the same point on the drawing unless it is changed with the origin command or at the time the drawing is retrieved.

ORTHOGONAL LINE - Draw a line vertically or horizontally.

Menu Command: ORTHO LIN
Short Form: OL
Keystroke Command: ALT-V

Point 1-n: Points the line is to be drawn through.

The Ortho Line command is used to draw a straight line or vector exactly horizontally or vertically. The command is used just like the Line command - enter ORTHO LIN, set 2 or more points, and press ENTER. To use the keystroke command, set the points and press ALT-V.

A line will be drawn through the points, but at each point the line will be adjusted so that it lies exactly horizontally or vertically, whichever is closer.

This command is useful when vertical and horizontal lines are being drawn with a mouse or digitizing tablet.

OVERLAY - Read an overlay from disk.

Menu Command: OVERLAY

Point 1: Lower left corner of overlay (optional).
Point 2: Lower right corner of overlay (optional).
Point 3: Upper left corner of overlay (optional).
Point 4: Upper right corner of overlay (optional).

The Overlay command retrieves a drawing from disk and displays it on the screen, but does not add that drawing to the current drawing - it is only displayed on the screen for information or scale purposes. The points used for this command are identical to those used in the Retrieve command:

Point 1: Point at which the lower left corner of the overlay is to be located.
Point 2: Point at which the lower right corner of the overlay is to be located.
Point 3: Point at which the upper left corner of the overlay is to be located.
Point 4: Point at which the upper right corner of the overlay is to be located.

None of these points are required. See the Retrieve command for further details.

This command retrieves a drawing and displays it on the screen. The drawing retrieved does not become part of the current drawing - it is only for display purposes.

An overlay will not be redrawn if a Zoom or Rotate command is used. To remove an overlay from the screen, you may redraw the current drawing using the Zoom command with a Zoom Factor of 1. This redraws the current drawing as is, leaving out the overlay.

An overlay is different from a layer. An overlay is a separate drawing retrieved with the Overlay command

for viewing purposes only. A layer is a subdivision of the current drawing that can be manipulated with the Layer command.

PAINT - Fill an area on the screen.

Menu Command: PAINT
Keystroke Command: P

Point 1: Location inside area to be painted.

The Paint command allows you to fill in or paint an area on the screen. To use the command, set a point inside the area to be filled, or painted. The area to be painted must be completely enclosed. Next, enter PAINT or press the P key.

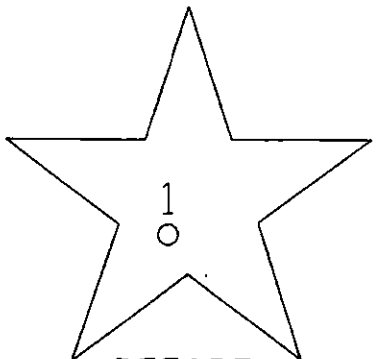
DesignCAD will fill the screen beginning at the point you set and continuing until it runs into a solid line. Figure 20 shows a star that has been painted using this command.

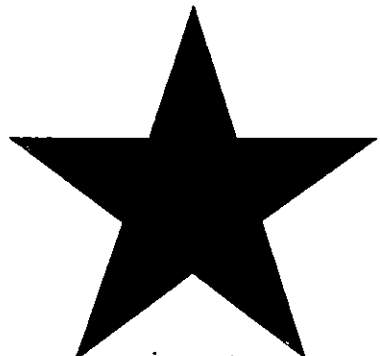
If the area you attempt to paint is not completely enclosed, DesignCAD will continue painting outside the area. Be sure the area to be painted is completely enclosed before you use the Paint command. The Paint command will paint areas only on the screen.

You may terminate the paint command at any time with the ESC key.

To erase a "painted" area, set a gravity point at one of the vertical edges of the area and enter ERASE (or press the E key).

THE PAINT COMMAND





THE PAINT COMMAND "PAINTS" OR FILLS AN AREA ON THE SCREEN.

FIGURE 20

PAN - Pan the drawing across the screen.

Menu Command: PAN

The Pan command is used to move or pan the drawing across the screen. To use the command, enter PAN. You will then be asked for the distance and direction of the pan.

Enter the distance you want the drawing to be moved, in Drawing Units. Then, enter the direction in which the DRAWING (not the screen window) is to be moved, in degrees. The drawing will be moved accordingly.

PARALLEL – Draw a parallel line.

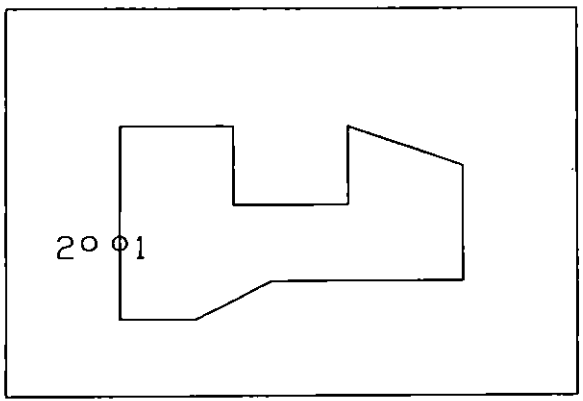
Menu Command: PARALLEL
Short Form: PARA
Keystroke: =

Point 1: Set point on original line.
Point 2: Location for parallel line.

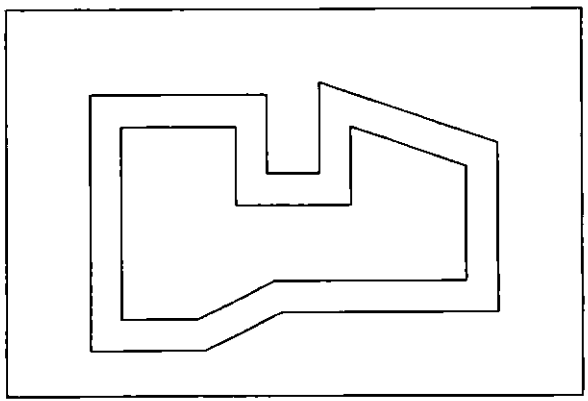
The Parallel command enables you to draw a line parallel to any other line, curve, circle, arc, ellipse, or elliptical arc.

To use the command, enter PARALLEL. Then set a point on the line to which a parallel is to be drawn, and a point at the location of the parallel line. To use the keystroke command, set the points and press the "=" key. A line parallel to the first line will then be drawn. See Figure 21.

PARALLEL COMMAND



BEFORE



AFTER

THE PARALLEL COMMAND IS USED TO DRAW A LINE PARALLEL TO ANOTHER LINE IN THE DRAWING. A PARALLEL CAN BE DRAWN TO A LINE, CURVE, CIRCLE, ELLIPSE, ARC, OR ELLIPTICAL ARC.

FIGURE 21

PERPENDICULAR - Draw a perpendicular line.

Menu Command: PERP

Point 1: Point from which perpendicular is drawn.
Point 2: Point on existing line.

The Perpendicular command is used to draw a line perpendicualar to another existing line. To use the command, enter PERP and set two points.

The first point is the point at which the perpendicular line is to start. Set the second point on the existing line to which the perpendicular is to be drawn. A new line will be drawn from the first point to the line designated by the second point, and will be perpendicular to that line.

PLOT - Output the drawing to the plotter.

Menu Command: PLOT
Keystroke Command: F6

This command outputs the current drawing to the plotter. To use the command, enter PLOT or press the F6 key. You will be asked for the following information:

1. Drawing Length.
2. Drawing Height.
3. Special Plotter Commands.

If you have specified the number of Drawing Units per inch on output, the appropriate size for your drawing will be displayed on the screen under Drawing Length and Height (provided it will fit on the plotter). Otherwise, the maximum plotter size will be displayed. These values can be changed by moving the cursor to the appropriate field and entering the new value.

If you want a particular command output to the plotter, you may enter it under the Special Plotter Commands field. For example, if you are plotting on transparencies, you may want to use the slower pen velocity. Enter that command for your plotter. (Commands for your particular plotter may be found in your plotter manual.)

After you made any necessary changes, press F1 to proceed with the plot. The plot may be terminated at any time with the ESC key.

A drawing may be plotted to a specific scale using the "Drawing Units per Inch on Output" parameter under the System Parameter command. See System Parameter command.

To specify the type of plotter you use, run the program SETUP. This program needs to be run only once.

POINT MOVE - Move a point in a line.

Menu Command: POINT MOVE
Short Form: PM
Keystroke Command: *

Point 1: Set point on point to be moved.
Point 2: New location for that point.

The Point Move command is used to move or delete a point in the drawing. To use the command to move a point enter POINT MOV. Set a point directly upon another point in the drawing, and a point at the location to which the first point is to be moved. Every command in the drawing containing the first point will be redrawn using the second point instead.

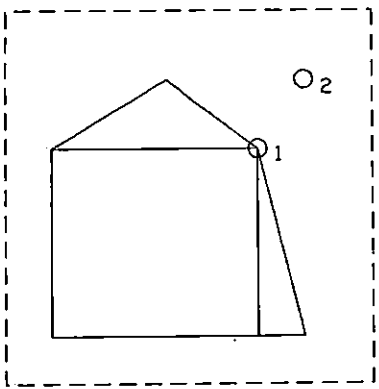
To use the keystroke command, set the points and then press the "*" key.

The Gravity command can be used to set the first point directly upon an existing point.

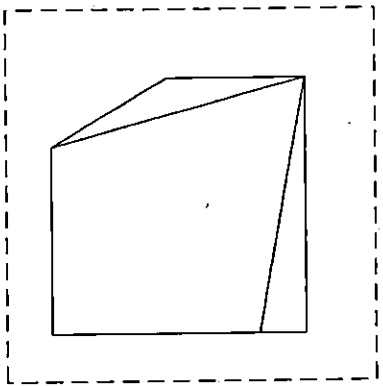
To use the Point Move command to delete a point, set the two points on top of the point to be deleted. Every command in the drawing using that point will be redrawn without that point.

Figure 22 shows an example of the Point Move command.

POINT MOVE



BEFORE



AFTER

THE POINT MOVE COMMAND IS USED TO MOVE OR DELETE AN INDIVIDUAL POINT WITHIN THE DRAWING. THE FIRST POINT IDENTIFIES THE POINT TO BE CHANGED. THE SECOND POINT IS THE DESTINATION OF THE FIRST POINT.

FIGURE 22

POINT SET - Set a point at the cursor location.

Keystroke Command: "0"

This command is used to set a point at the cursor location. To use the command, press the 0 (Zero) key. A point will be set at the cursor. To "unset" the point, press ESC.

This command can also be performed by pressing the first button on a mouse or digitizer.

POINT SET, ABSOLUTE COORDINATES - Set a point using absolute coordinates.

Menu Command: POINT XY
Short Form: PXY
Keystroke Command: :

This command is used to set a point by specifying the X and Y coordinates of that point. To use the command, enter POINT XY or press the ":" key. You will be asked for the X and Y coordinates of the point. A point will then be set at that location.

The X coordinate is the horizontal location of the point. The Y coordinate is the vertical location of the point. The X and Y coordinates are in Drawing Units. The X-Y position of the cursor can be displayed by specifying format 3 with the Status Line command.

POINT SET, POLAR COORDINATES - Set a point using polar coordinates.

Menu Command: POINT PLR
Short Form: PP
Keystroke Command: ;

This command is used to set a point by specifying the distance and angle (polar coordinates) of the new point from the last point set (menu command entry) or from the current cursor position (keystroke entry). To use the command, enter POINT PLR or press the ";" key.

You will be asked for the distance and angle from the cursor set to the new point. Enter the distance in the current Drawing-Units and the angle in degrees. A point will then be set at that distance and angle from the last point set. If no points are set, the distance will be calculated from the current cursor position.

For example, to use this command to draw a line of length 10 at an angle of 45 degrees, set the first point using the zero key on the numeric keypad. Then enter POINT PLR, and enter a distance of 10 and an angle of 45 degrees. The second point will be set at that location. Next, enter the LINE command (or press the V key). A line of length 10 will then be drawn between the points at a 45 degree angle.

The distance can be entered using feet and inches by separating the feet and inches with a ('). The " character may optionally follow the inches:

8'6" is equivalent to 8.5

The angle can be entered in degrees, minutes, and second by separating the degrees from the minutes with D, and the minutes from seconds with M. S may optionally follow the seconds:

30D30M30S is equivalent to 30.508333

If the point polar command is used with the semicolon key, then the distance is calculated from the current cursor position. If the command is entered from the menu or command line, then the distance is calculated from the last point set.

POINT SET, RELATIVE COORDINATES - Set a point using relative coordinates.

Menu Command: POINT REL
Short Form: PR

The Point Relative command is used to set a point relative to the current cursor position. The position of the new point is given in horizontal (X) and vertical (Y) distances from the last point set. If no points have been set, the distances will be calculated from the current cursor position.

To use this command, enter POINT REL. You will then be asked for the relative X offset and the relative Y offset. The relative X offset is the horizontal distance from the cursor to the new point location. The relative Y offset is the vertical distance from the cursor to the new point location.

After these values are entered, the cursor will move to the new point location and a point will be set there.

The distance can be entered using feet and inches by separating the feet and inches with a ('). The " character may optionally follow the inches:

8'6" is equivalent to 8.5

POLYGON - Draw a regular polygon.

Menu Command: POLYGON
Short Form: POLY

Point 1: First point on a side of the polygon.
Point 2: Second point on the polygon side.

The Polygon command draws a regular polygon, that is, a convex shape in which each side is the same length. To use the command, enter POLYGON. Then enter two points that will be one side of the polygon.

You will then be asked for the number of sides the polygon is to have. Enter 3 for a triangle, 4 for a square, 5 for a pentagon, etc. The polygon will then be drawn.

For example, to draw a regular octagon (a stop sign shape), set the second point to the right of the first, and enter 8 when asked for the number of sides. The octagon will be will be drawn using the two points for its bottom.

PRINT - Output the drawing to the printer.

Menu Command: PRINT
Keystroke Command: F5

The DesignCAD Print command is capable of producing extremely high resolution drawings on ordinary dot matrix printers. To use the Print command, enter PRINT or press the F5 key. You will be asked if you want to proceed with the print command and if you need to save your drawing.

The Print command is actually a separate program from DesignCAD, so the current drawing must be saved before entering the Print command or it will be lost. After entering the Print command, any drawing on disk may be printed.

If you are using floppy disks, you will be asked to replace the DesignCAD program disk with the Print disk. If you have 3.5" disks, you do not need to replace the program disk with the print disk.

When the Print command begins, you will be given the following options:

1. Print a DesignCAD Drawing.
2. Print Several DesignCAD Drawings.
3. Display DesignCAD Drawings on Disk.
4. Specify Layers to be Printed.
5. Terminate the Print Command and Return.
6. Stop the Program.

Option 3 is like the DesignCAD Directory command.

Option 4 is like the Layer command, except that there is no Current Layer with the Print command - every layer is either Active or Inactive. Active layers are printed, Inactive layers are not.

Option 5 returns to the main program DesignCAD program, and option 6 terminates the program and returns to DOS.

Option 1 is used to actually print a drawing. Option 2 can be used to print more than one drawing at a time.

When you select option 1, you are asked for the drawing name. After the drawing is retrieved, you will be asked for the information below. When option 2 is selected, the information below is entered before the drawings are retrieved, and each drawing is printed using the same set of parameters.

1. DENSITY
This is the printer resolution: normal, high, or very high. (If your printer does not support multiple resolution or densities, you will not be asked for this.) Higher resolution output looks better, but takes longer to print.

2. REQUESTED DRAWING UNITS PER INCH
This makes it easy to make scale drawings on your printer. For example, if one Drawing Unit represents a foot in your drawing, and you want your output to be scaled at 1 inch = 4 feet, then specify 4 drawing units per inch. Items 3 and 4 will be automatically scaled according to your drawing (provided that the requested scale will fit on your printer).

In order for the Drawing Units and the Units per Inch on output to be used by the print command, they must have been saved by the System Parameter command (ALT-Q) either with the drawing or as program defaults.

If you do not want your drawing scaled automatically, enter 0 for the Drawing Units per Inch.

The size of your drawing in drawing units is displayed at the bottom of the screen (unless you are

printing multiple drawings). If you have entered a number too small for "Drawing Units per Inch on Output", the drawing may not fit on the printer and a warning will be displayed on the screen.

3. PRINTING AREA WIDTH
This is the width of the printing area on your printer. It corresponds to the output drawing height, unless the drawing is rotated 90 degrees (see #5 below). The maximum paper width is limited by the printer's capabilities. This is usually 8" for a printer with a 10" carriage or a little over 13" for a printer with a 15" carriage.

4. PRINTING AREA LENGTH
This is the length of the printing area on your printer. It corresponds to the output drawing width, unless the drawing is rotated 90 degrees (see #5 below). The Printing Area Length can be greater that one page.

5. ROTATE DRAWING 90 DEGREES?
The drawing normally comes out sideways on the page. If you want the drawing to come out with the drawing width the same direction as the printer width, enter Y.

6. CHARACTER FONT
This is the character font to be used in the printer output. Font 1 is a simplex, straight line font. Font 2 is a simplex curved font. Font 3 is a Roman type font.

7. NUMBER OF COPIES TO BE PRINTED
More than one copy can be printed at a time by specifying the number.

The Printing process can be terminated at any time be pressing the ESC key.

QUARTER CIRCLE - Draw a quarter circle.

Menu Command: QUART CIR
Short Form: QC
Keystroke Command: (

Point 1: Beginning of quarter circle.
Point 2: Ending point of quarter circle.

The Quarter Circle command is used to draw a quarter-circle between two points. To use the command, enter QUART CIR and set a point at the beginning and ending of the quarter circle. To use the keystroke command, set the points and then press the "(" key.

A quarter-circle, or 90 degree arc, will be drawn from the first point clockwise to the second point.

Although the Arc command can be used to draw quarter-circles, this command is easier to use in many cases.

REPEAT SECTION COPY - Copy a section again.

Menu Command: REPT COPY
Short Form: RC
Keystroke Command: "

Point 1: Location for new copy of section.

The Repeat Section Copy command is used after the Section Copy command to copy the same section again. The command is used in the same manner as the Section Copy command, except that the first two points that define the section to be copied are omitted.

To use this command, enter REPT COPY and set 1 to 4 points, or set the points and press the " key. The first point is the location to which the lower left corner of the section is to be copied. The second point, if used, is the location of the lower right corner of the section. The second point can be used to determine the size and angle of the new section.

The third point is the location of the upper left corner of the new section, and the fourth point is the location of the upper right corner of the section. The third and fourth points can be used to slant the section or to give it an isometric view with perspective.

RETRIEVE - Read a drawing or symbol from disk.

Menu Command: RETRIEVE
Short Form: LOAD
Keystroke Command: F9

Point 1: Lower left location for drawing (optional).
Point 2: Lower right location for drawing (optional).
Point 3: Upper left location for drawing (optional).
Point 4: Upper right location for drawing (optional).

This command is used to retrieve a drawing from disk and display it on the screen. It allows you to specify the location, size, and angle for the drawing.

With the Retrieve command, you can use up to 4 points. The first point specifies the location of the drawing. The lower left corner of the drawing will be positioned at the first point.

The second point specifies the size and angle of the drawing. The lower right corner of the drawing will be positioned at the second point. If only one point is used, the drawing will be drawn the size and angle at which it was saved.

The third point specifies the slant of the drawing. The upper left corner of the drawing will be located at the third point. If the third point is not used, the drawing will not be slanted.

The fourth point is the point at which the upper right corner of the drawing is to be located. If this point is not specified, the drawing will be located in a parallelogram defined by the first 3 points.

If no points are used, the drawing will be retrieved at the same size and location from which it was

saved. The relative size and location of the drawing, which may have been modified by the Zoom command, is taken into consideration when the drawing is retrieved.

To use the menu command, enter RETRIEVE and set 0 to 4 points, followed by ENTER. To use the keystroke command, set the points and then press F9. You will be asked for the drawing name. Enter the drawing name as you did when it was saved.

The origin or primary handle of the drawing is ordinarily the lower left corner of the drawing. However, one or two handles can be specified when the drawing is saved.

When a handle has been specified with the Save command, then the first or primary handle will be placed at the first point instead of the lower left corner of the drawing.

If the secondary handle is specified with the Save command, then the drawing being retrieved is scaled and rotated so that the secondary handle is located on the second point set for the Retrieve command.

If the handles have been saved with a drawing, then the third and fourth points are not used when the drawing is retrieved.

If the drawing is on a drive or subdirectory other than the one DesignCAD is on, you may precede the drawing name by the drive specification or path name:

```
B:filename
```

The drawing will then be drawn on the screen. See Figure 23.

If you cannot remember the name of the drawing, you can use the Directory command to display the names of all the drawings stored on disk.

If you slant a drawing (with the third and fourth points) containing arcs, circles, curves, or ellipses, the curved lines will not be slanted proportionally. If you want to slant commands with curves, you can use the ALT key with the curved-line keystroke commands.

For example, if you are creating a drawing that will later be slanted, you should use ALT-O instead of O to draw a circle. This causes the circle to be broken down into a series of line commands. The disadvantages of this technique are that the resolution of the curve will not increase when you output to a plotter or zoom in on the drawing, and a line command usually takes more space than its normal counterpart.

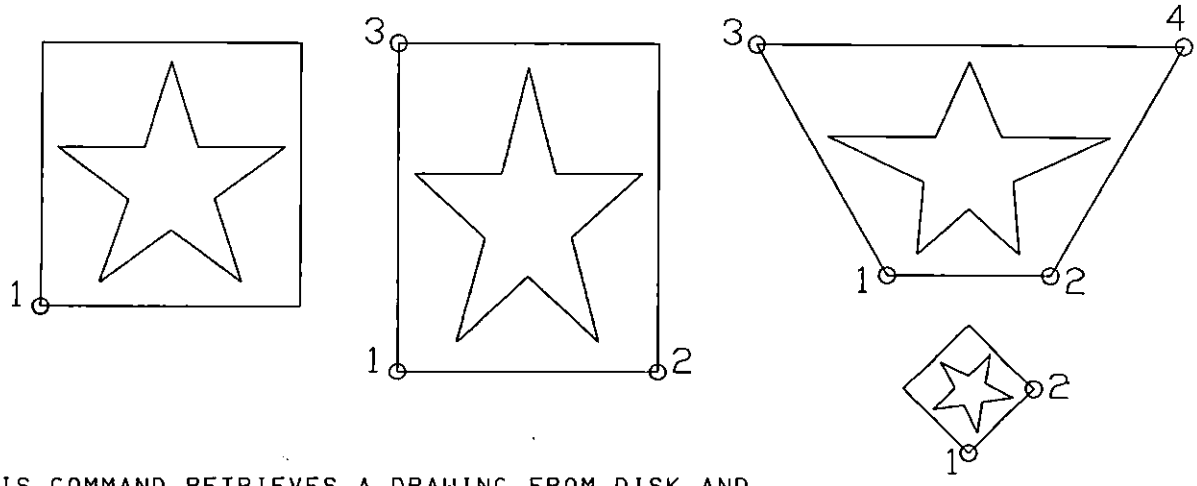
It is possible to merge or combine drawings by retrieving one drawing on top of another.

The retrieve command can be performed automatically when you start DesignCAD by entering

```
DC filename
```

where "filename" is the name of the drawing you want to load.

THE RETRIEVE COMMAND



THIS COMMAND RETRIEVES A DRAWING FROM DISK AND DRAWS IT ON THE SCREEN. UP TO 4 POINTS MAY BE USED, ALTHOUGH NONE ARE REQUIRED.

FIGURE 23

ROTATE - Rotate the drawing

Menu Command: ROTATE
Short Form: R
Keystroke Command: R

Point 1: Center of rotation (optional).
Point 2: New location for point 1 (optional).

The Rotate command allows you to rotate the entire drawing. To use the command, enter ROTATE or press the R key. You will be asked for the Rotation Angle. Enter the number of degrees you want your drawing to be rotated. (0 leaves it the same, 45 moves it counter-clockwise 45 degrees, etc.)

You may set 0, 1, or 2 points with this command. The first point is the point about which the drawing is rotated. If you do not set any points, the drawing will be rotated about the center of the screen.

The second point, if specified, is the location to which the first point is to be moved. If only 1 point is specified, that point will remain in the same location.

A drawing can also be rotated by using four points with the Zoom command. See the Zoom command for additional information.

To rotate the drawing back to its original angle, press ENTER when asked for the rotation angle.

RUBBER BAND BOX - Enable the rubber band box.

Menu Command: RUB BOX
Short Form: RB
Keystroke Command: [

This command displays a box or rectangle, using the cursor and the last point that was set, to define the opposite corners of the rectangle. The rectangle moves around with the cursor until the next point is set or until the next drawing command is performed.

To activate the Rubber Band Box, enter RUB BOX or press the "[" key. The Rubber Band Box will be automatically turned off when the second point is set. The Rubber Band Box can also be turned off by entering RUB BOX (or pressing "[") again.

RUBBER BAND LINE - Enable the rubber band line.

Menu Command: RUB LINE
Short Form: RL
Keystroke Command: -

This command places a line from the cursor to the last point that has been set (if any). It works as if a rubber band were stretched from the previous point to the cursor. The line moves around with the cursor until the next point is set or until the next drawing command is performed.

To activate the Rubber Band Line, enter RUB LINE or press the "-" key. To turn it off, enter RUB LINE (or press "-") again.

SAVE - Save the drawing.

Menu Command: SAVE
Keystroke Command: F10

Point 1: Primary handle (optional).
Point 2: Secondary handle (optional).

The Save command is used to save a drawing onto disk. To use this command, enter SAVE or press the F10 key. You will be asked for an output file name. Entér the name under which you want to store your drawing. The name must be 1 to 8 characters long, using letters or numbers. The drawing will then be stored onto disk.

One or two points may be set before entering SAVE to specify the origins or handles of the drawing. The handles of the drawing are the reference points at which the drawing can be retrieved. (See the Retrieve command.) If no point is set, the handles of the drawing will be the lower left corner and the lower right corner of the drawing.

If you want to store the drawing on a drive or subdirectory other than the one DesignCAD is on, you may precede the drawing name by the drive specification or path name:

B:filename

If you are editing an existing drawing and want to save it under the same name, the "=" key may be used to specify the original drawing name. When asked for the output file name, press the "=" key. The original drawing name will be displayed. If that is the correct name, press ENTER.

The file name under which a drawing is stored is "filename.DC2", where "filename" is the name of the drawing. This command will store the entire drawing,

even though parts of it may not lie on the screen. To delete a drawing from the disk, use the Directory command. A drawing may also be deleted using the MSDOS Erase command:

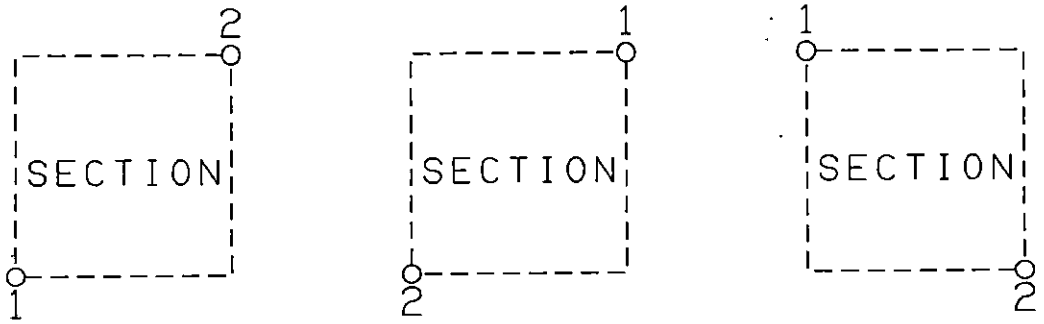
ERASE filename.DC2

where "filename" is the name of the drawing to be deleted. This command is executed only after you exit DesignCAD.

If the Zoom command has been used on a drawing, the copy of the drawing saved on disk is scaled back to its original size and moved to its original location. The drawing on the screen is not changed.

The drawing size may be changed permanently by using the Static Zoom command (ALT-Z) or by retrieving the drawing at its desired size (by setting points with the Retrieve command) and then saving it.

DEFINING A SECTION



TO DEFINE A SECTION, SET A POINT IN OPPOSITE CORNERS OF A RECTANGLE. SECTIONS ARE USED WITH SECTION MOVE, COPY, SAVE, AND DELETE COMMANDS.

FIGURE 24

SECTION COPY - Copy a Section.

Menu Command: SEC COPY
Keystroke Command: N
Keystroke Command: ALT-N (for line breaking)

Point 1: Corner of the section.
Point 2: Opposite corner of the section.
Point 3: Location for lower left corner of copy.
Point 4: Location for lower right corner (optional).
Point 5: Location for upper left corner (optional).
Point 6: Location for upper right corner (optional).

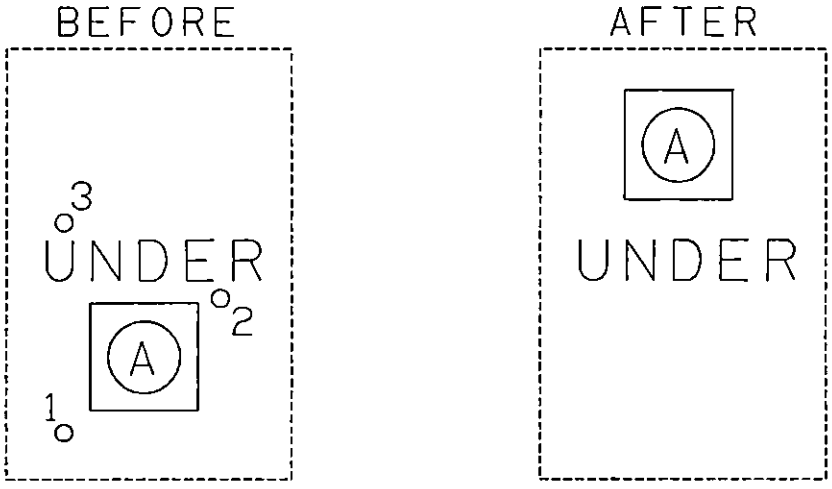
The Section Copy command is used to copy a section of the drawing from one location to another. A section is part of the drawing contained in a rectangle defined by two points in opposite corners.

To use the menu command, enter SEC COPY. Then set 3 to 6 points, followed by ENTER. Define the section by setting two points in opposite corners of a rectangle. See Figure 24. Next, set a third point at the position to which you want the lower left corner of the section to be moved.

To use the keystroke command, set the points and press the N key.

You may optionally set a fourth, fifth, or sixth point to define the position of the lower right corner, upper left corner, and upper right corner of the section. The fourth point, if used, is the location of the lower right corner of the section. The fourth point can be used to determine the size and angle of the new section.

SECTION COPY COMMAND



THIS COMMAND COPIES A SECTION FROM ONE LOCATION TO ANOTHER. THE FIRST TWO POINTS DEFINE THE SECTION. THE THIRD POINT IS THE LOCATION FOR THE LOWER LEFT CORNER OF THE SECTION.

FIGURE 25

After the points have been set, press ENTER. Any line or entity (e.g. line, curve, circle, arc, text) within the section defined will be copied to the new location.
The fifth point is the location of the upper left corner of the section, and the sixth point is the location of the upper right corner of the section. The fifth and sixth points can be used to slant the section or to give it an isometric view with perspective.

This Section Copy command moves the lines or entities located entirely within the section or rectangle. If a command lies partly within and partly outside the section, it will not be moved.

To "cut off" or break lines at the border of the section, the keystroke command ALT-N can be used instead of N. The Cutoff menu command can also used before the Section Delete command.

This command is like the Section Move command except it does not delete the original section. The Repeat Section Copy command can be used to copy the section again without setting the two points to define the section.

SECTION DELETE - Delete a Section.

Menu Command: SEC DEL
Keystroke Command: D
Keystroke Command: ALT-D (for line breaking)

Point 1: Corner of the section.
Point 2: Opposite corner of the section.

The Section Delete command is used to delete or erase a section of the drawing. A section is a rectangle defined by two points in opposite corners.

To use the menu command, enter SEC DEL. Then define the section by setting two points in opposite corners of a rectangle, and press ENTER.

To use the keystroke command, set the points and then press the D key.

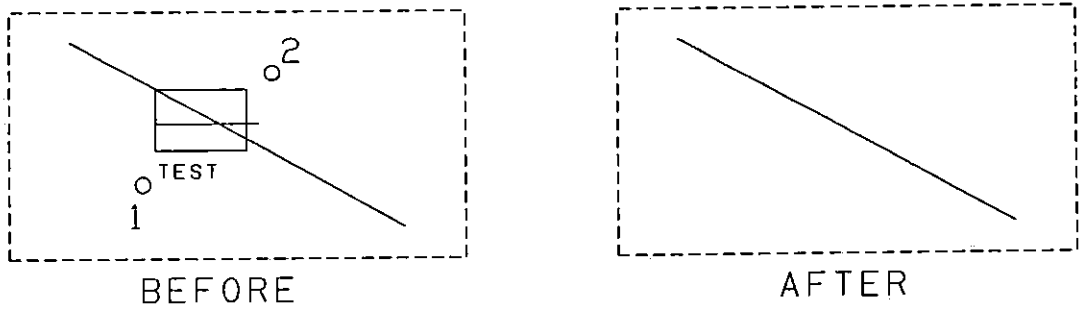
After the points have been set, any line or drawing entity (e.g. line, curve, circle, arc, text) within the section defined will be deleted from the drawing. See Figure 26.

This command deletes the lines or entities located entirely within the section or rectangle. If a command lies partly within and partly outside the section, it will not be deleted.

To "cut off" or break lines at the border of the section, the keystroke command ALT-D can be used instead of D. The Cutoff menu command can also used before the Section Delete command.

The Oops command can be used to replace items accidentally deleted with the Section Delete command.

SECTION DELETE COMMAND



THE SECTION DELETE COMMAND IS USED TO DELETE OR ERASE A SECTION OF THE DRAWING. ANY DRAWING COMMAND COMPLETELY WITHIN THE WILL BE ERASED.

FIGURE 26

SECTION MOVE - Move a Section.

Menu Command: SEC MOVE
Keystroke Command: M
Keystroke Command: ALT M (for line breaking)

Point 1: Corner of the section.
Point 2: Opposite corner of the section.
Point 3: New location for lower left corner.
Point 4: Location for lower right corner (optional).
Point 5: Location for upper left corner (optional).
Point 6: Location for upper right corner (optional).

The Section Move command is used to move a section of the drawing from one location to another. A section is a rectangle defined by two points in opposite corners.

To use this command, enter SEC MOVE. Then define the section by setting two points in opposite corners of a rectangle. Next, set a third point at the position to which you want the lower left corner of the section to be moved.

After the points have been set, press ENTER. Any line or entity (e.g. line, curve, circle, arc, text) within the section defined will be moved to the new location.

To use the keystroke command, set the points and then press the M key.

You may optionally set a fourth, fifth, or sixth point to define the position of the lower right corner, upper left corner, and upper right corner of the section. The fourth point, if used, is the location of the lower right corner of the section. The fourth point can be used to determine the size and angle of the new section.

The fifth point is the location of the upper left corner of the section, and the sixth point is the location of the upper right corner of the section. The fifth and sixth points can be used to slant the section or to give it an isometric view with perspective.

This Section Move command moves the lines or entities located entirely within the section or rectangle. If a command lies partly within and partly outside the section, it will not be moved.

To "cut off" or break lines at the border of the section, the keystroke command ALT-M can be used instead of M. The Cutoff menu command can also used before the Section Delete command.

The Section Move command is like the Section Copy command except that the original section is erased, and only the new copy remains.

SECTION SAVE - Save a Section.

Menu Command: SEC MOVE
Keystroke Command: X
Keystroke Command: ALT X (for line breaking)

Point 1: Corner of the section.
Point 2: Opposite corner of the section.

The Section Save command is used to save a section of the drawing onto disk. A section is a rectangle defined by two points in opposite corners.

To use this command, enter SEC SAVE. Then define the section by setting two points in opposite corners of a rectangle.

A third or fourth point can be used to define the alternate origins or handles of the saved file. After the points have been set, press ENTER.

You will then be asked for an output file name. Enter a name, 1 to 8 characters long, using letters or numbers. Any line or entity (e.g. line, curve, circle, arc, text) within the section defined will then be saved on disk.

To use the keystroke command, set the points and then press the X key.

If you want to store the section on a drive or subdirectory other than the one DesignCAD is on, you may precede the file name by the drive specification or path name:

B:SECNAME

The file name under which the section is stored is secname.DC2", where "secname" is the name of the section. To erase a section from the disk, use the directory command.

This command saves the lines or entities located entirely within the section or rectangle. If a command lies partly within and partly outside the section, it will not be saved.

To "cut off" or break lines at the border of the section, the keystroke command ALT-X can be used instead of X. The Cutoff menu command can also used before the Section Delete command.

SEMI-CIRCLE - Draw a Semicircle

Menu Command: SEMI CIRC
Short Form: SC
Keystroke Command:)

Point 1: Beginning point of semicircle.
Point 2: Ending point of semicircle.

This command draws a semi-circle between two points. To use the command, enter SEMI CIRC and set two points. A semi-circle, or 180 degree arc, will be drawn from the first point clockwise the second point.

To use the keystroke command, set the points and then press the ")" key.

Although the Arc command can be used to draw semi-circles, this command is easier to use in many cases.

SKETCH - Enable the sketch mode.

Menu Command: SKETCH

The sketch mode is a mode in which a line is drawn wherever the cursor moves.

To enable the Sketch mode, enter SKETCH. If your display has 640 or better horizontal resolution, then word SKETCH will appear at the top of the screen on the status line while the Sketch mode is enabled.

To begin drawing a line, press the first button on the mouse or digitizing tablet. To stop the line, press the first button again.

To turn off the Sketch mode, enter SKETCH again or any other DesignCAD command.

SNAP GRID - Turn the Snap Grid on or off.

Menu Command: SNAP GRID
Short Form: SNAP
Keystroke Command: ALT-G

The Snap Grid command enables or disables the Snap Grid. The Snap Grid is an imaginary grid placed over your drawing. Each point you set with the 0 key (or the first mouse or digitizer button) will be located exactly on one of the grid points.

The distance between the points on the Snap Grid is the snap grid size. The snap grid size is set using the Drawing Parameter command (Q key).

The Snap Grid does not affect points set with the Gravity Point (. key) or Coordinate Point (; or : keys) commands.

To turn the Snap grid on or off, enter SNAP GRID or press ALT-G.

STATUS LINE FORMAT - Set the status line format.

Menu Command: STATUS LN
Short Form: SL
Keystroke Command: J

The Status Line command is used to specify the format of the DesignCAD status line. The status line is the line of information at the top of the screen.

To use the command, enter STATUS LN or press the J key. You will then be asked for the status line format (1-3). The three formats contain the following information:

1. a. Number of Points Set.
   b. Current Layer.
   c. Current Zoom Factor.
   d.* Current Color.
   e.* SNAP, if the Snap Grid is on.
   f.* MACRO, if a Macro is being made.
   g.* SKETCH, if the Sketch mode is on.

2. a. Number of Points Set.
   b. Distance from the cursor to the last point.
   c. Angle of the cursor from the last point.
   d.* Delta X, or horizontal distance from the cursor to the last point.
   e.* Delta Y, or vertical distance from the cursor to the last point.

3. a. Number of Points Set.
   b. X coordinate of the cursor.
   c. Y coordinate of the cursor.

(* Items are not displayed on the 320x200 resolution IBM Color Graphics Adapter.)

The values in formats 2 and 3 are updated as the cursor is moved about the screen. The distance and

X,Y coordinates are measured in Drawing Units specified by the Units command. A distance of eight horizontal "dots" or pixels is used as one Drawing Unit if the units have not been defined.

Status line formats 4 and 5 are similar to formats 2 and 3, but distances are displayed in feet and inches instead of decimal fractions.

There are two types of angle measurement that can be used with DesignCAD - mathematical and geographical. The type of angle is selected under the System Parameter command (SYSTEM, or ALT-Q). The angle is measured in degrees, according to the method of angle measurement being use.

With mathematical angles, the degrees progress counter-clockwise, with 0 degrees at the right (3 o'clock) and 90 degrees straight up (12 o'clock). With Geographical angles, the degrees progress clockwise, with 0 degrees straight up (12 o'clock) and 90 degrees to the right (3 o'clock). See the System Parameter command.

STRETCH - Stretch a Section.

Menu Command: STRETCH
Short Form: S
Keystroke Command: S

Point 1: Corner of the section.
Point 2: Opposite corner of the section.
Point 3: Point to which section is to be stretched.

The Section Stretch command is used to stretch a section of the drawing from one location to another. A section is a rectangle defined by two points in opposite corners.

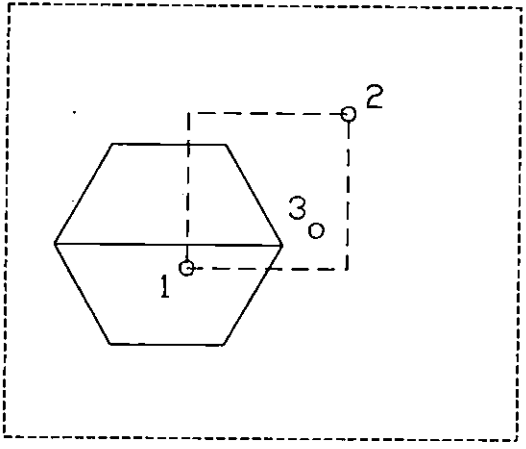
To use this command, enter STRETCH, and define the section by setting two points in opposite corners of a rectangle. Next, set a third point at the position to which you want the lower left corner of the section to be stretched.

After the points have been set, press ENTER. Any drawing entity (e.g. line, curve, circle, arc, text) within the section defined will be moved to the new location. If a line or entity lies partly within and partly outside the section, it will be stretched between the new location and the old. See Figure 27.

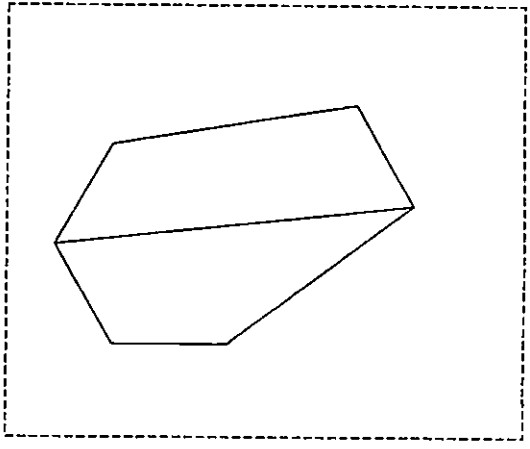
To use the keystroke command, set the points and then press the S key.

STRETCH COMMAND

BEFORE



AFTER



THIS COMMAND STRETCHES A SECTION. THE FIRST TWO POINTS DEFINE THE SECTION, AND THE THIRD POINT IS THE LOCATION TO WHICH THE LOWER LEFT CORNER OF THE IS TO BE STRETCHED.

FIGURE 27

SYSTEM PARAMETER - Enter System Parameters.

Menu Command: SYSTEM
Keystroke Command: ALT-Q

The System Parameter command is used to specify DesignCAD system parameters, such as character font, precision, and drawing units per inch.

To use the command, enter SYSTEM or press ALT-Q. A screen as shown in Figure 28 will be displayed. To change one of the parameters, move the cursor to that location and enter the new value. The system parameters are described below.

1. CHARACTER FONT
You can select alternate character fonts with DesignCAD. After a new font is selected, all the text on the screen is erased and re-written in the new character font.

Font 1 is a simplex, straight line font. Font 2 is a simplex curved font. Font 3 is a Roman type font.

The Vector Text keystroke command (ALT-T key) can be used instead of the Text command (TEXT menu command, or T keystroke command) to prevent text from being affected by this command.

For example, to use text in both fonts 1 and 2 in a drawing, you can draw all the text to be displayed in character font 1 using the Vector Text command, change to font number 2, and then draw all the text for character font 2. The text drawn using the Vector Text command will not be changed to character font 2.

PRODESIGN II SYSTEM PARAMETERS

CHARACTER FONT (0-9):	1
DRAWING UNITS PER INCH ON OUTPUT:	0.000
MATHEMATICAL OR GEOGRAPHICAL ANGLES (M/G):	M
PRECISION (FRACTIONAL DIGITS) (0-7):	3
SOUND ON? (0=OFF, 1=ERROR ONLY, 2=ON):	1
FILL WIDE LINES? (Y/N):	N
DISPLAY ATTRIBUTES? (Y/N):	N
MANIPULATE CURRENT LAYER ONLY? (Y/N):	N
COMMAND MENU ACTIVE? (Y/N):	Y
DIMENSION FORMAT (1-6):	1
DYNAMIC DIMENSIONING ENABLED? (Y/N):	Y
SAVE SYSTEM PARAMETERS WITH DRAWING? (Y/N):	N

USE ARROW KEYS TO SELECT AN ITEM TO BE CHANGED.
PRESS "F1" TO USE NEW VALUES (BUT NOT SAVE THEM).
PRESS "F2" TO USE NEW VALUES AND SAVE THEM AS DEFAULTS.
PRESS "ESC" TO ABORT.

FIGURE 28

Character font number 0 may be used to "hide" the text. When font number 0 is used, each line of text in the drawing is displayed as a rectangle, although the text is still part of the drawing. Since the text is not redrawn when the Zoom and Rotate functions are performed, the Zoom and Rotate commands are faster when character font 0 is used.

2. UNITS PER INCH ON OUTPUT
The units per inch on output specifies the number of Drawing Units to be output per inch on paper. This makes it convenient to make scale drawings.

For example, to make a scale drawing at 1 inch equals 4 feet, set one drawing unit equal to one foot in the drawing using the Units command. Then set the units per inch on output equal to 4.

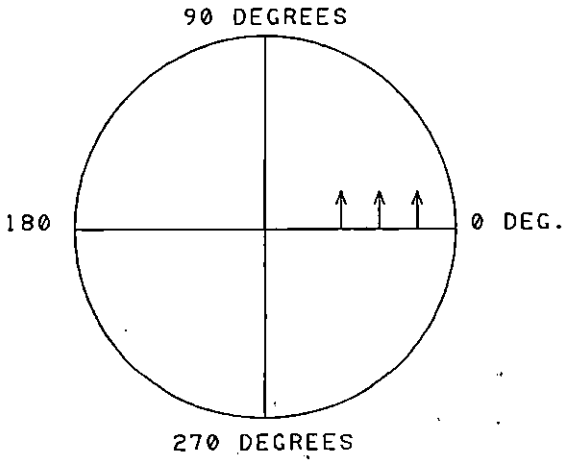
If the Metric option was selected during DesignCAD setup, then this item will be "Units per cm on output" instead of "Units per inch on output".

3. MATHEMATICAL OR GEOGRAPHICAL ANGLES
This determines the location and direction of angle measurement. With Mathematical angles, 0 degrees is to the right (or 3 o'clock) and the degrees progress counter-clockwise. With Geographical angles, 0 degrees is straight up (or 12 o'clock) and the degrees progress clockwise. See Figure 29.

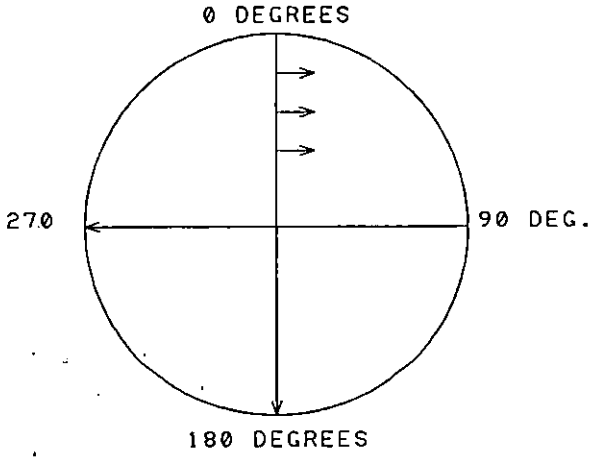
4. PRECISION
The precision is the number of digits to the right of the decimal point for values on the status line and in the dimension annotation. This value does not affect the way the drawing is maintained or saved.

ANGLE MEASUREMENT

MATHEMATICAL



GEOGRAPHICAL



PRODESIGN II ANGLES CAN BE MEASURED USING EITHER MATHEMATICAL OR GEOGRAPHICAL CONVENTIONS. THE TYPE OF ANGLE IS SPECIFIED IN THE "SYSTEM PARAMETER" COMMAND.

FIGURE 29

5. SOUND ON?
This determines whether a "click" will be made after each point is set and after a command has been performed.
6. FILL WIDE LINES?
Wide lines can be left hollow for faster processing, or can be filled or "painted".

7. DISPLAY ATTRIBUTES?
This allows you to specify whether Attributes will be displayed on the screen. An Attribute is a line of text entered with the Attribute command. For more information, see the Attribute command.

8. MANIPULATE CURRENT LAYER ONLY?
This item determines whether the Section and Block commands are used with and affect the current layer only, or all visible layers.

9. COMMAND MENU ACTIVE?
The command menu can be disabled or enabled by entering N or Y here.

10. DIMENSION FORMAT (1-6)
Any of the following dimension formats can be selected. See Figure 30.

1 - Decimal, number on the line.
2 - Feet/Inches, decimal to .1 inch.
3 - Feet/Inches, fraction to 1/32"
4 - Inches, fraction to 1/64"
5 - Decimal, number above the line, slashes instead of arrows.
6 - Decimal, number above the line.

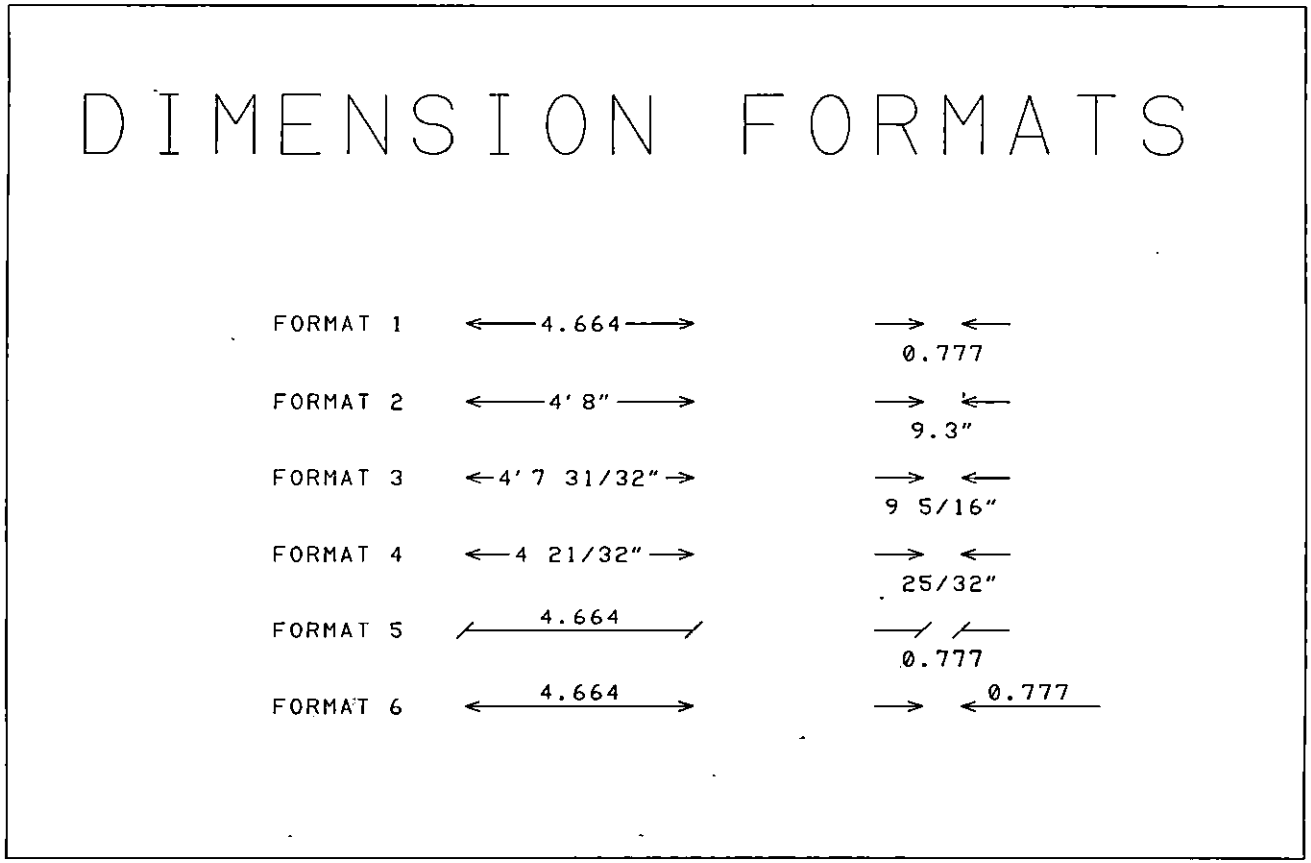

DIMENSION FORMATS
FORMAT 1
4.664
0.777
FORMAT 2
4' 8"
9.3"
FORMAT 3
4' 7 31/32"
9 5/16"
FORMAT 4
4 21/32"
25/32"
FORMAT 5
4.664
0.777
FORMAT 6
4.664
0.777

FIGURE 30

11. DYNAMIC DIMENSIONING ENABLED?
If Y is entered here, the dimensions will be "dynamic". This means that the dimensioning information will be changed to correspond to the current units of measurement. If dynamic dimensioning is disabled, the dimensions will be drawn as lines and text, and can be edited as such.

12. SAVE PARAMETERS WITH DRAWING?
If Y is entered here, the pertinent System Parameters, Drawing Parameters and Drawing Units information will be saved with the drawing. If N is entered, the those parameters will not be saved with the drawing.

If the parameters are saved with the drawing, they will become the new system and drawing parameters whenever that drawing is retrieved, unless there is already a drawing on the screen.

The following parameters are saved with the drawing:

1. Cursor increment.
2. Display Grid information.
3. Snap Grid information.
4. Units per inch on output.
5. Default Text Size and Angle.
6. Precision.
7. Fillet Radius.
8. Drawing Units.
9. Dimension format and type.

The System Parameter command can be terminated three ways:

F1 - Use the system parameters but do not save them to disk.
F2 - Save the parameters as program defaults. (Use the system parameters every time the

program begins.)
ESC - Return and don't change anything.

Drawing Units, in addition to several of the the Drawing and System Parameters, are saved when the Drawing Parameter command is terminated with the F2 key. The parameters saved as defaults are:

1. Cursor increment.
2. Display Grid information.
3. Snap Grid information.
4. Precision.
5. Character Font.
6. Default Text Size and Angle.
7. Drawing Units per inch on output.
8. Fillet Radius.
9. Attribute Display on or off.
10. Drawing Units size.
11. Menu on or off.
12. Mathematical or Geographical angles.
13. Fill Wide Lines on or off.
14. Sound Level.
15. Edit current section only - on or off.
16. Drawing origin.
17. Save paramters with drawing.
18. Dynamic Dimensioning enabled.
19. Dimension format.

TANGENT - Draw a tangent to a circle or arc.

Menu Command: TANGENT
Short Form: TAN

Point 1: End point of tanget line.
Point 2: Set on circle to which tangent is drawn.

The Tangent command is used to draw a line from a point tangent to a circle, ellipse, arc, or elliptical arc.

To use this command, enter TANGENT. Then set a point at which the line is to begin, and a point on or near the circle (or ellipse, arc, or elliptical arc). A line will then be drawn from the first point to the circle (or ellipse, arc, or elliptical arc), and tangent to the circle. See Figure 31

TANGENT-2 - Draw a tangent to 2 circles or arcs.

Menu Command: TANGENT-2
Short Form: TAN2

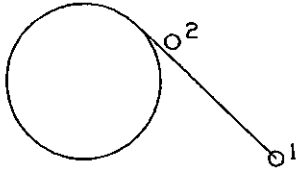
Point 1: Set on 1 circle to which tangent is drawn.
Point 2: Set on other circle.

The Tangent-2 command is used to draw a line between two circles or arcs, tangent to both.

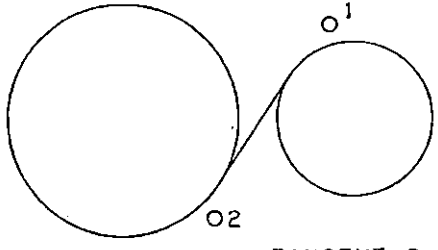
To use this command, enter TANGENT-2. Then set a point near each of the two circles or arcs the tangent line is to connect.

There are four tangent lines that can connect two circles. the tangent line is selected that is nearest the two points set. See Figure 31.

TANGENT COMMANDS



TANGENT



TANGENT-2

THE TANGENT COMMAND DRAWS A LINE FROM A POINT TANGENT TO A CIRCLE OR ARC. THE TANGENT-2 COMMAND DRAWS A LINE TANGENT TO TWO CIRCLES.

FIGURE 31

TEXT - Draw Text.

Menu Command: TEXT
Keystroke Command: T
Keystroke Command: ALT-T

Point 1: Lower left corner of text.
Point 2: Lower right corner of text (optional)
Point 3: Upper left corner of text (optional)
Point 4: Upper right corner of text (optional)

The Text command is used to write text onto the screen. With the Text command, you may set 1 to 4 points.

To use this command, enter TEXT or set the points and then press the T key.

If only one point is used, the size and angle of the text is determined by the default text size and the default text angle. These values are set using the Drawing Parameter command.

The second point is used to specify the length and angle of the text. The lower right corner of the text will be positioned at the second point.

The third point, if used, is the point at which the upper left corner of the text is to be located. The fourth, if used, is the point at which the upper right corner of the text is to be located. (Text in character font #2 should not be slanted.)

After all the points have been set, press ENTER. You will be asked to enter "TEXT: " at the top of the screen. The message you enter will then be displayed on the screen.

Text can be centered about a point by setting the first two points in the same location and then

pressing ENTER. The text will be centered about the two points. The size and angle of centered text is determined by the default text size and angle. Figure 32 gives several examples of the Text command.

The ALT-T keystroke command can be used to draw text and convert it to the actual lines used to make each letter. This allows other character fonts to be loaded without affect the text on the screen, but it requires more storage for the text.

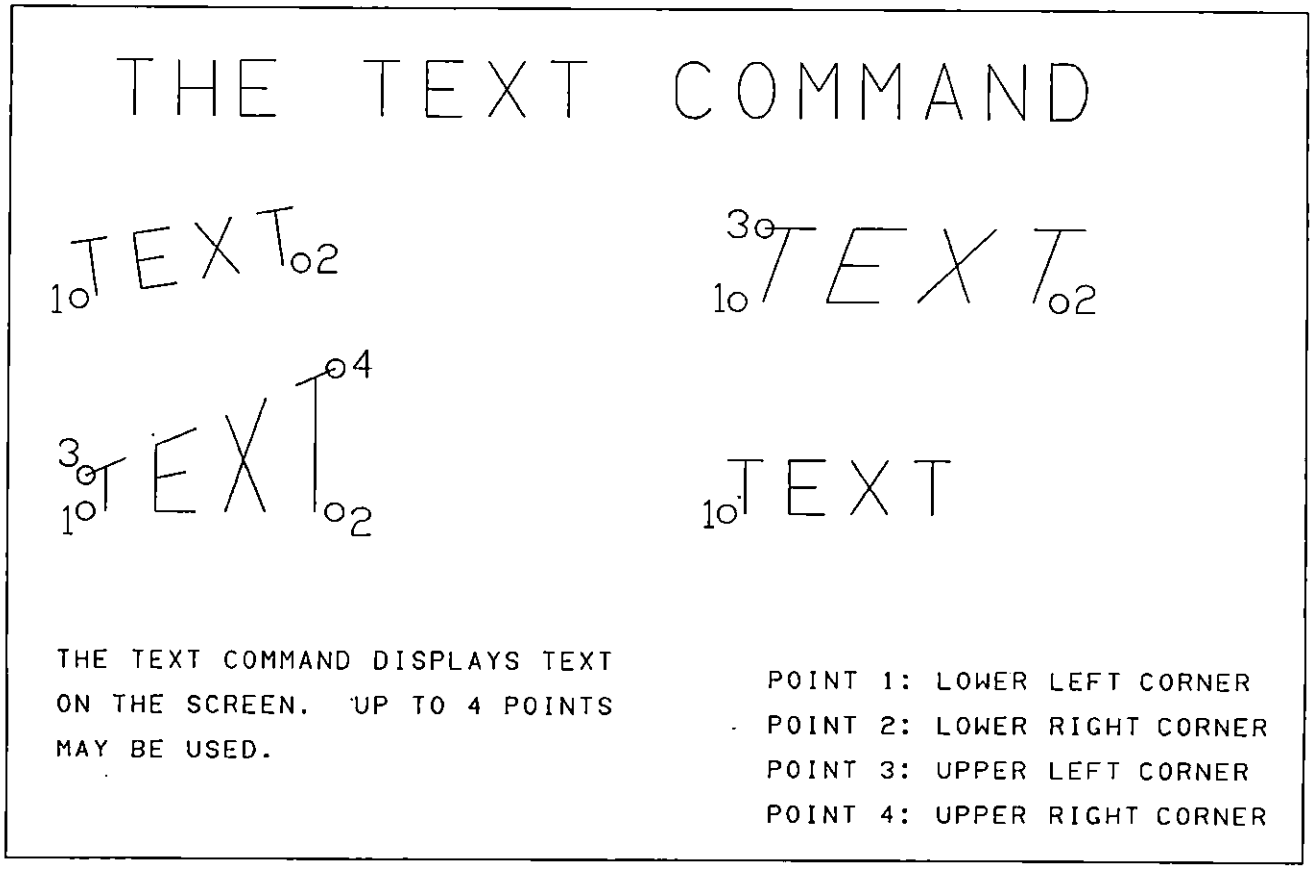

THE TEXT COMMAND
1 TEXT 2
3 1 TEXT 2
3 1 TEXT 2 4
1 TEXT
THE TEXT COMMAND DISPLAYS TEXT ON THE SCREEN. UP TO 4 POINTS MAY BE USED.
POINT 1: LOWER LEFT CORNER
POINT 2: LOWER RIGHT CORNER
POINT 3: UPPER LEFT CORNER
POINT 4: UPPER RIGHT CORNER

FIGURE 32

TEXT ARC - Draw text in an arc between two points.

Menu Command: TEXT ARC
Short Form: TA

Point 1: Beginning point of arc.
Point 2: Any point on the arc except endpoints.
Point 3: Ending point of arc.

The Text Arc command is used to draw a line of text in an arc. To use the command, enter TEXT ARC. Then set three points that lie on the arc - the beginning, a point other than endpoints, and the ending point of the arc.

You will then be asked for the text. The line of text is drawn in an arc from the first point through the second point to the third point.

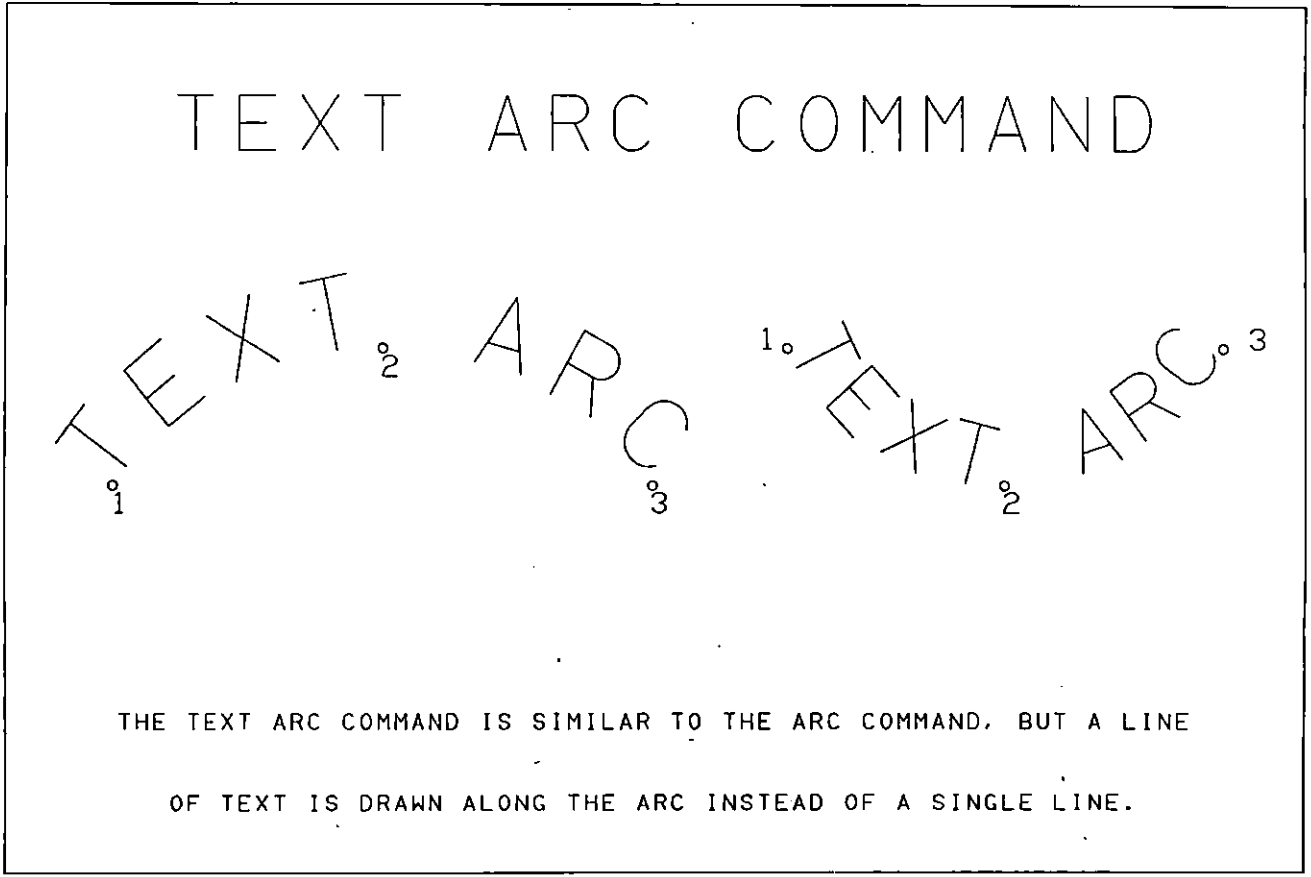

TEXT ARC COMMAND
TEXT ARC
1 TEXT 2 ARC 3
THE TEXT ARC COMMAND IS SIMILAR TO THE ARC COMMAND, BUT A LINE
OF TEXT IS DRAWN ALONG THE ARC INSTEAD OF A SINGLE LINE.

FIGURE 33

TRIM - Trim a line.

Menu Command: TRIM
Short Form: TR

Point 1: Set on endpoint to be moved.
Point 2: New location for the endpoint.
Point 3: Set on line to be trimmed (optional).

The Trim command is used to shorten or lengthen a line. It is similar to the point move command, but the trimmed line maintains exactly the same angle as before.

To use the command, set a point on the endpoint of the line to be changed, and a point at the new location for the endpoint. The line will be shortened or lengthened according to the location of the second point. The angle of the line remains the same.

If the second point is not precisely "in line" with the line, then the endpoint will be moved as close as possible to the second point, while still maintaining the angle of the line.

A third point can be set to further specify the line to be moved. If a third point is set, then the closest line to that point is the line that will be trimmed. Otherwise, the line with the closest endpoint to the first point set will be trimmed.

UNDO - Undo the last line or entity drawn.

Menu Command: UNDO
Keystroke Command: ESC

The Undo command is used to erase or undo the last line or entity drawn. To use the command, enter UNDO or press the ESC key. The last command that was drawn will be erased from the screen. The Undo command can be repeated to erase more than one line or entity.

The Undo command will not undo a Zoom, Rotate, Section, or Block command.

UNITS - Specify units of measurement.

Menu Command: UNITS
Keystroke Command: U

Point 1: Beginning of measured distance.
Point 2: Ending of measured distance.

The Units command is used to set the units of measurement for the drawing. These units of measurement are called Drawing Units. Drawing Units can be feet, miles, centimeters, inches - any unit of measurement that makes sense for the drawing.

To use the Units command, enter UNITS, and set two points a certain distance apart. You will then be asked the number of units that the distance between the points represents. Enter that distance.

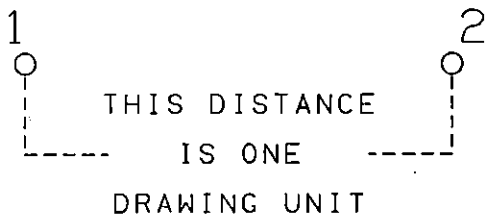
To use the keystroke command, set the two points and then press the U key.

For example, if there is an "8 foot" line in a drawing and you want the Drawing Units to be 1 foot, set points at the end points of the line, and answer "8" when asked for the number of units.

Figure 34 gives an example of the Units command.

The Drawing Units are used with several of the parameters under the Drawing Parameter command, such as the Grids and the Cursor Step Size. Dimension information and positions on the status display are given in Drawing Units. Drawing Units can also be used to determine the output size of the drawing on a printer or plotter.

THE UNITS COMMAND



TO SET THE DRAWING UNITS, SET TWO POINTS A CERTAIN DISTANCE APART IN THE DRAWING. THEN ENTER THE NUMBER OF DRAWING UNITS THAT DISTANCE REPRESENTS. IN THIS EXAMPLE, YOU WOULD ENTER "1". THE DRAWING UNITS ARE USED AS THE UNITS OF MEASUREMENT FOR MANY PRODESIGN II COMMANDS.

FIGURE 34

VECTOR DRAWING COMMANDS

ALT-A - Vector Arc
ALT-C - Vector Curve
ALT-O - Vector Oval
ALT-T - Vector Text

These commands draw on the screen exactly as the corresponding drawing commands. The only difference is that the commands are broken down into vectors and saved as such. This is sometimes useful when the drawing (or parts of the drawing) will be slanted or inverted, because the curved commands cannot always be slanted or distorted.

WIDTH - Enter the line width or thickness.

Menu Command: WIDTH
Keystroke Command: W

This command specifies the width or thickness of the lines used in the drawing commands. To use the command, enter WIDTH or press the W key. You will be asked for the line width.

Enter the desired line width: 0 for thin lines, larger values for thicker lines. For example, enter .5 to make the line .5 units thick, .2 for .2 units thick, etc.

A wide line can be hollow or filled, depending on the "Fill Wide Lines" parameter setting of the System Parameter command.

ZOOM - Zoom in or out.

Menu Command: ZOOM
Keystroke Command: Z

Point 1: Point to be zoomed into (optional).
Point 2: New location for point 1 (optional).
Point 3: Alternate reference point (optional).
Point 4: New location for point 3 (optional).

The Zoom command is used to change the size of the current drawing. To use the command, enter ZOOM. You will be asked for the "Zoom Factor". The Zoom Factor is the relative size you want the drawing to become: 2 for twice as large, .5 for half as large, etc.

A zoom factor of 0 may be specified to scale the drawing so that it fits as large as possible into the screen. To return your drawing to its original size and location, press ENTER when asked for the zoom factor.

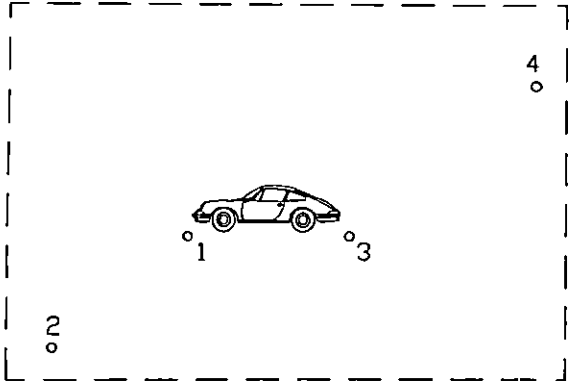
One, two, or four points may be used with the Zoom command. The first point is the point into which you want to "Zoom". The second point is the location to which the first point will be moved.

If no points are specified, the center of the screen will be used for the zoom location. If only one point is specified, then that point in the drawing will be moved to the center of the screen.

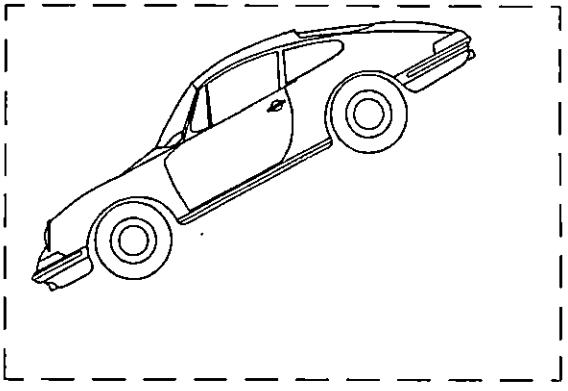
Four points may be used with the Zoom command to perform a combination Zoom and Rotate. The first two points function as they would normally. The first point will be moved to the second point on the screen.

THE ZOOM COMMAND

USING FOUR POINTS



BEFORE



AFTER

YOU CAN USE FOUR POINTS WITH THE ZOOM COMMAND TO DETERMINE THE ZOOM AND ROTATION FACTORS.

FIGURE 35

The third and fourth points function similarly. The third point will be moved to the fourth point. The drawing will be scaled and rotated so that the first and third points fit into the second and fourth points. See Figure 35.

When the Zoom command is used with a digitizing tablet, the points on the digitizing tablet remain consistent with the points on the drawing. The larger you zoom, the larger the cursor movement becomes on the digitizing tablet.

For example, if you draw a line on the screen using the digitizing tablet, and Zoom in by a factor of 2, the line becomes twice as large. However, the end points of the line remain in the same place on the digitizing tablet.

When the drawing is zoomed to become larger or smaller, the Cursor Step Size and the Display Grid remain the same size relative to the screen, not to the drawing. When you return to the Drawing Parameter command after a zoom, the Cursor Step Size and Display Grid may change in value, but they are the same size on the screen.

The Static Zoom keystroke command (ALT-Z) can be used to change the size of the drawing with respect to the digitizing tablet.

ZOOM STATIC - Static Zoom

Menu Command: ZOOMSTAT
Keystroke Command: ALT-Z

Point 1: Point to be zoomed into (optional).
Point 2: New location for point 1 (optional).
Point 3: Alternate reference point (optional).
Point 4: New location for point 3 (optional).

The Static Zoom command is exactly like the Zoom command, except that the internal zoom factor and drawing position do not change, even though the drawing size and location does.

The command can be used to change the size or position of a drawing. For example, if you have a drawing that is too small when read from disk, you can use the Static Zoom command with a zoom factor of 2, and then save the drawing. The drawing will then be twice its original size when read from disk with the Retrieve command.

The Static Zoom command can also be used to align a drawing on the screen with a drawing on a digitizing tablet. For more information on using this command with a digitizing tablet, see the section below entitled "USING A DIGITIZING TABLET".

ZOOM WINDOW - Zoom with a window.

Menu Command: ZOOMW
Short Form: ZW

Point 1: Corner of zoom window.
Point 2: Opposite corner of window.

The Window Zoom command is another form of the Zoom command. To use the command, enter ZOOMW. Next, set two points for the "zoom window". The zoom window is a rectangle or window defined by two points in opposite corners.

The portion of the drawing inside the zoom window will be magnified so that it fills the screen. The location and magnification of the zoom depend on the size and location of the zoom window.

USING DESIGNCAD

This section describes some of the more advanced techniques that are useful in using DesignCAD.

SETTING POINTS

DesignCAD uses points set on the screen to define locations for the drawing commands. These points may be set using several methods:

Point Set Command - "0"
First button on mouse or digitizer.
Gravity Point - ".", GRAVITY
Point Set, Absolute Coordinates - ":", POINT XY
Point Set, Polar Coordinates - ";", POINT PLR
Point Set, Relative Coordinates - POINT REL
Set Point at Nearest Line - LINESNAP
Set Point at Nearest Midpoint - MIDPOINT
Set Point at Intersection - INTERSECT

The Point Set command sets a point at the current cursor location. It is the simplest of these. This is the same as pressing the first button on the mouse or digitizing tablet.

The Gravity Point command is used to set a point exactly upon the nearest point. With the keyboard, this saves keystrokes by moving the cursor immediately to its destination (when another point is the destination) without having to "zero in" on the point. With a digitizing tablet or a mouse, it allows you to find the spot without worrying about a steady hand.

When you zoom, move, and rotate a drawing, the points and lines may end up lying a little bit off the center of the dots on the screen. The cursor, however, remains exactly on top of the dots or pixels

on the screen. A point set on top of another using the Point Set command (0 Key) may in fact be a fraction of a dot off, although it will appear to be correct. When the drawing is zoomed in, the difference can be seen. The use of the gravity point prevents this error from occurring.

For example, draw a diagonal line on the screen. Zoom by a factor of .3 . Then, using the 0 key and the Line command, make a triangle out of the diagonal line. Then zoom in on one of the end points of the original line by a factor of 10. (Set a point on one of the end points and then Zoom.) You will see that the lines of the triangle do not actually meet. While this was not apparent on the screen, it would be seen on printer or plotter output.

The Point Set, Abolute Coordinates command (":" Key or POINT XY) can be used to set a point at an exact location on the screen. The coordinates of the current cursor position can be displayed using the Statue Line Format. The coordinates are entered in the current units of measurement.

The Point Set, Relative Coordinates command (POINT REL) is similar to the POINT XY command, but the X and Y values are relative instead of absolute. This means that the X and Y values are added to the X and Y values of the current cursor position.

The Point Set, Polar Coordinates command (";" Key or POINT PLR) can be used to set a point a certain distance and angle from the last point that was set. This allows you to set points at precise locations without keeping track of the absolute screen coordinates.

The LINESNAP command sets a point at the nearest point of the nearest line. This command is similar to the Gravity Point command, but it goes to the nearest line instead of the nearest point.

The MIDPOINT command sets a point at the midpoint of the nearest line. If the line has more than one segment, it sets a point at midpoint of the nearest line segment.

The INTERSECT command is used to set a point at the intersection of two lines, circles, ellipses, arcs, or elliptical arcs. (Any combination of these can be used except for two ellipses or elliptical arcs.) This command requires two points to be set, one on each of the intersecting lines. Those two points will be erased, and a new point will be set on the intersection. If points are set before the INTERSECT command, they will not be lost.

USING A DIGITIZING TABLET

DesignCAD supports several different digitizing tablets (also called graphics tablets and digitizers).

To set up DesignCAD to use your digitizing tablet, run the program SETUP. If the program has not been run before, you will be asked for printer and plotter information as well as information on your digitizing tablet. Otherwise, select option 3 and enter the number for your digitiziner tablet. You will then be asked for the communications information.

After the digitizing tablet information has been entered, choose option 6 to align the digitizer menu.

ALIGNING THE DIGITIZER MENU

With your DesignCAD package, there is a rectangular piece of cardboard with the DesignCAD commands on it. This is the digitizer menu. The digitizer menu can be affixed to the top of your digitizing tablet to allow you to enter DesignCAD commands directly from the digitizing tablet.

The digitizer menu must be aligned precisely in order for the commands to be in their proper locations. To do this, select option 6 from the SETUP program.

You will be asked to move the cursor or stylus until position 0,0 is displayed on the screen. When you have found this location, place the lower left corner of the digitizer menu there, and press ENTER.

If no numbers appear at the bottom of the screen, then the DesignCAD is not communicating with your digitizer. Check the switch settings on the digitizer, and make sure DesignCAD is configured for your digitizer.

Next, the location of 0,0 changes to the position for the lower right corner of the digitizer menu. Move the cursor or stylus again to the 0,0 location as shown on the screen. Place the lower right corner of the digitizer menu at this location, and press ENTER.

USING THE DIGITIZING TABLET

The digitizing tablet can be used to move the cursor, set points, and enter DesignCAD commands. To move the cursor, merely move the stylus or cursor around the digitizing tablet. The cursor will move about the screen in a corresponding manner.

Your stylus or digitizer cursor will have from 1 to 15 (or more) buttons. Up to 15 buttons can be used to enter DesignCAD commands. Each button corresponds to a different command:

Button 1. Point Set
Button 2. Gravity Point
Button 3. Line
Button 4. Curve
Button 5. Ellipse
Button 6. Fillet
Button 7. Chamfer
Button 8. Line
Button 9. Section Copy
Button 10. Repeat Section Copy
Button 11. Point Move
Button 12. Section Delete
Button 13. Erase
Button 14. Escape
Button 15. Oops

These commands may also be entered from the keyboard or digitizer menu.

The area of the screen accessed by the digitizing tablet may be varied using the Zoom command (Z key). When DesignCAD is initialized, the area on the screen accessed by the digitizing tablet is the largest area in the shape of the digitizing tablet that can be displayed on the physical screen.

If you want the digitizing tablet area to cover the entire screen, set a point in the lower left corner of the screen and Zoom with a factor of 2. If you Zoom even more, the digitizing tablet will access areas outside the screen. It is possible to use the digitizing tablet to draw outside the screen. Of course, when this is done you cannot see what you are drawing on the screen.

The Zoom command will not change the relationship of the drawing in the computer and the digitizing tablet - it only changes the size or location of the drawing on the screen.

If you need to change the location of the drawing in the computer with respect to the digitizing tablet, you can use the Static Zoom command (ALT-Z key) to zoom instead of the Zoom command. The ALT-Z command works like the normal Zoom command except that the digitizing tablet remains constant with the screen instead of the drawing in the computer.

Suppose, for example, that you are making a drawing from an existing drawing on 11x17 paper, but you are using an 11x11 digitizing tablet. You can attach the paper to the digitizing tablet to draw the left portion of the existing drawing. Then, you will need to shift the paper on the digitizing tablet to access the right side of the drawing.

After the paper has been shifted, it will no longer be aligned with the drawing in the computer. You can use the ALT-Z command to correct the situation. Select a point on the drawing that is on both the drawing in the computer and the paper on the digitizing tablet. Set a point at that location on the drawing in the computer. Then set a point at that location on the paper on the digitizing tablet.

Now select a second point, not too close to the first point, that also appears on both the drawing in the computer and on the paper on the digitizing tablet. Just as before, set your third point at that location on the drawing in the computer. Then set the fourth point at that location on the paper on the digitizing tablet.

Finally, press ALT-Z to zoom and rotate the drawing so that it is aligned with the paper on the digitizing tablet. The four points used are:

1. Source point 1 on the screen.
2. Destination point 1 on the digitizing tablet.
3. Source point 2 on the screen.
4. Destination point 2 on the digitizing tablet.

Your digitizing tablet may have either a stylus (a pen-like device) or a cursor (also called a mouse). A stylus can be depressed onto the digitizing tablet in order to set a point. If the stylus has a button or barrel switch, this may be used to set a point with gravity. If not, the gravity command on the digitizer menu may be used. This command is described in the command section of this manual.

USING A MOUSE

A mouse can be used with DesignCAD to position the cursor, set points, draw lines, and enter commands from the command menu.

A mouse usually has 2 or 3 buttons. If you have a two button mouse, you can simulate the third button by pressing the two buttons at the same time. The buttons on the mouse can be used to enter the following commands:

Button 1: Point Set
Button 2: Gravity Point
Button 3: Line

The mouse can also be used to enter commands from the command menu. To do this, move the cursor to the right side of the screen with the mouse. A box will appear around one of the commands on the menu. Move to the desired command by moving the mouse up and down. Select the command by pressing the first button on the mouse.

If you are using a mouse, the Gravity Point command should be entered either from the second button on the mouse or with the keystroke command ("." key). If you select the menu command, the cursor will move to the right of the screen before the command is executed, making it difficult to snap to the correct location.

ERASING PARTS OF YOUR DRAWING

There are several ways to erase parts of a drawing:

Undo Command.
Erase Command.
Section Delete Command.
Section Delete with Line Breaking.
Block Delete Command.

The Undo command (ESC or UNDO) can be used to erase the last line or entity drawn. It can be repeated to erase more than one command. If a command is erased in error, the Oops command ("!" or OOPS) can be used to reverse an Undo. Like the Undo command, the Oops command can be repeated. If anything has been drawn or the drawing has been "zoomed" since the last Undo, the Oops command cannot be used.

The Erase command (E or ERASE) is used to selectively erase lines or commands in the drawing. A point is set on the line to be erased. The point can be on an endpoint of the line, or it can be on (or 1 dot or pixel away from) the line itself. If the point lies directly on an endpoint, the command will function faster in a large drawing. The Gravity point is very useful for this.

The Section Delete command (D key) allows an entire section of the drawing to be erased at one time. A section is made up of all the lines or entities completely contained within the rectangle defined by two points set in opposite corners. (See the Section Delete Command). Any command that lies completely within the section is erased.

Section Delete with Line Breaking will actually break lines and curves that cross the section boundaries and delete the portions of those lines inside the section. This command is particularly useful in making an opening in a line.

The Block delete command erases a previously defined Block (See the section below on Blocks).

When a line that intersects another line is erased on the drawing, a small portion of the intersected line is erased from the screen. This portion is not erased from the drawing itself - it is only erased from the display. The drawing can be regenerated to repair these line breaks using the Zoom command (Z key) with a Zoom factor of 1.

By the same token, it is NOT possible to erase a line by drawing over the line and pressing the ESC key. While this does erase the line from the screen, it is not removed from the actual drawing and will reappear when the drawing is regenerated with a Zoom, Rotate, or Save and Retrieve.

USING ORTHO VECTORS

If you use a mouse or digitizing tablet, you may find it useful to use the Ortho Vector command to draw horizontal and vertical lines. With a mouse or digitizer, it is often difficult to determine when the cursor is in line with a point horizontally or vertically. The Ortho Vector command (ALT-V) can be used to draw a line at exact horizontal or vertical angles (whichever is closer).

When copying a drawing from a digitizing tablet, however, the Ortho Vector command may cause problems if the drawing on the tablet is not exactly aligned. If the drawing on the tablet is tilted slightly, a horizontal line in the drawing on the tablet will not correspond to a horizontal line in the drawing on the computer.

ZOOMING

The Zoom function of DesignCAD is a very powerful feature. You can make your drawing larger than the screen, or smaller than a single dot on the screen, without losing any part of the drawing. In addition, you can use the Zoom to "pan" or move your drawing across the screen.

To use the Zoom command to make your drawing larger or smaller, press the Z key and enter the zoom factor. The zoom factor is the relative size you want the drawing to be. For example, to make a drawing twice as large, enter 2 for the zoom factor. To double the size again, zoom with a factor of two. To reduce the drawing to half its original size, zoom with a factor of .125 (1/8). The absolute zoom factor is displayed on the status line.

If no points are set with the Zoom command, the center of the screen will be used as the point to zoom into (or out of). To zoom into a particular location on the drawing, set a point at that location. To zoom into a location and move it to another location on the screen, set a second point at the destination location.

If you only want to move the drawing one direction or another, set a point at the "from" location and a point at the "to" location and then zoom with a factor of one. The entire drawing will be moved or "panned" to the new location.

The PAN command can also be used to pan the drawing. This command moves the drawing a certain distance and direction, as specified.

To replace the drawing at its original size and location, a null zoom factor can be used with the Zoom command. (Press ENTER when asked for the zoom factor).

To make the drawing as large as possible on the screen, use a zoom factor of 0.

A digitizing tablet will "follow" the Zoom command. As you Zoom in, the digitizer will move the cursor the same distance on the drawing (and farther on the screen) for a given distance on the tablet.

The Static Zoom command (ALT-Z key) can be used to prevent the digitizer from "following" the Zoom, and also to keep the absolute zoom factor (displayed on the status line) from changing. This can be useful in aligning a drawing on the digitizing tablet to a drawing in the computer. It can also be used to alter the default drawing size - a drawing is Zoomed back to an absolute factor of 1 when it is saved.

The Zoom Window command (ZOOMW) can be used to zoom into a certain section of the drawing. With this command, a window or section is designated on the screen, and that area is scaled up so that it fills the screen as nearly as possible.

RETRIEVING DRAWINGS

Drawings may be retrieved from disk either with or without points. If the Retrieve command (F9 or RETRIEVE) is used without points, the drawing will be displayed at its original size and location. Points may be set with the Retrieve command to specify the location, size, and angle of the drawing.

One or two "handles" can be stored with a drawing. If 1 or 2 points were set when the drawing was saved, then the drawing will have 1 or 2 reference points or handles for retrieval.

When a drawing with handles is retrieved, the first handle is placed at the first point used with the Retrieve command and the second handle is placed at the second point. This is very useful with symbols.

More than one drawing can be retrieved and merged into a single drawing. This is done by retrieving the first drawing, and then retrieving the second drawing on top of it. The Drawing Units of the second drawing are matched to those of the drawing already on the screen.

USING SYMBOLS

Your particular type of drawing may require the frequent use of common figures or symbols. For example, an engineering flow diagram may use a valve symbol several times.

DesignCAD makes it easy to create and use such symbols. A symbol is created as an independent DesignCAD drawing. It can be created at a larger size to allow detail and then reduced using the Zoom command.

Suppose you want to create a valve symbol. First, draw the valve on the screen at a relatively large size. (Begin with a blank DesignCAD screen.) Next, reduce the valve to its normal size using the ALT-Z command. The normal Zoom command could be used here, but ALT-Z will keep the drawing from being expanded to its original size when it is saved. Of course, the symbol can be drawn at its normal size and saved without the Zoom. All the symbols you will use can be created this way.

To use a symbol in your drawing, set a point at the location you want the lower left corner of the symbol to be positioned, and retrieve the symbol with the F9 key.

If a symbol needs to be retrieved based on a point other than the lower left corner, then the symbol can be saved with a reference point, or handle. Then the symbol can be retrieved based on the handle rather than the lower left corner.

A secondary handle can be used to replace the lower right corner of the drawing as a reference point.

The handles are specified by setting points at the handle locations when the Save command (F10 Key) is performed.

If you need to retrieve the symbol at an angle, or if you need to alter the size of the symbol, you can use two points with the Retrieve command.

After a symbol is drawn on the screen, it can be moved or copied with the Section Move command (M key) or Section Copy command (N key), or it can be manipulated with any of the Block commands.

It is sometimes useful to make a drawing containing only a row or two of frequently used symbols. This drawing, or symbol menu, can then be retrieved to an out-of-the way place on the screen, and symbols can be copied from the menu into the main drawing.

There are many symbols included with DesignCAD at no extra charge. These symbols can be retrieved onto a drawing with the Retrieve command just as the symbols you create.

USING TEXT

The DesignCAD Text command is capable of displaying text in several different manners. The simplest way to draw text is to use a single point with the Text command. The text will then be drawn at the default size and angle beginning at that point. The default text size and angle can be modified using the J key.

If you want to center the text, set two points in the same location. The text will be centered about that point at the default size and angle.

To locate text between two points, set those two points with the Text command. The text will be drawn from the first point to the second point. The size of the text depends on the distance between the points. The angle of the text depends on the angle of the two points.

A third point can be used to slant the text - the upper left corner of the text is located at the third point. A fourth point can be used to specify the location of the upper right corner of the text. These two points may cause the text to be slanted and stretched. Some character fonts may be distorted improperly when this is done. To get around this, the text can be drawn using only two points with the ALT-T command and stretched to the desired position with the Section Move command.

EDITING YOUR DRAWING

One of the primary advantages of CAD systems is the ability to edit and manipulate drawings. DesignCAD provides complete editing facilities for this.

The Section Move command (M key) can be used to move sections of your drawing around. Similarly, the Section Copy (N key) is used to duplicate sections of the drawing. Section Move and Section Copy transport whole lines or commands.

The ALT-M and ALT-N command (Section Move and Copy with Line Break) can be used to break a line or curve on the section boundary. These commands function exactly like the Section Move and Copy commands, but they will "cut off" lines and curves that lie partly within and partly outside the section.

If a portion of a line or command needs to be modified, the Change Point ("*" key) command can be used. This command allows you to actually move a point in the drawing. Every line or command using the point is "stretched" to accommodate the new location of the point. A point can also be deleted using the Change Point command. This is useful in erasing one end of a line.

To open or erase a section of a line, the ALT-D command can be used.

The Block commands also provide extensive editing capabilities, as described in the following section.

USING BLOCKS

The DesignCAD Block commands provide very flexible drawing and editing capabilities. There are several block commands:

BLK ADD	Add an entity to a Block.
BLK CIRC	Duplicate a Block in a circle.
BLK COLOR	Change the color of a Block.
BLK DEL	Delete a Block.
BLK HANDL	Set or move the Block's Handle.
BLK INSRT	Copy a Block one time.
BLK LAYER	Change the layer of a Block.
BLK MIR	Make a mirror image of a Block.
BLK MOVE	Move a Block.
BLK REPT	Duplicate a Block a number of times.
BLK RESET	"Undefine" a Block.
BLK ROTAT	Rotate a Block.
BLK SAVE	Save a Block to disk.
BLK SET	Define a block.
BLK ZOOM	Zoom a block.

The Block Set command defines a block to be inside a window or rectangle. Single drawing entities can be added to the block using the Block Add command. The Block Set command is the only Block command with both a menu and keystroke command. All other Block commands are strictly menu commands.

The Block Reset and Block Handle command can be used to define a block without including commands in a window. This makes it possible to make a block out of a single (or few) line or drawing entity.

The Block Insert command can be used to copy and optionally rotate and scale a block. The block can be positioned so that its two reference points or "handles" fit between two points. To move a block, use Block Insert followed by Block Delete.

The Block Repeat command duplicates a block a number of times. The copies of the block can be spaced any distance and any direction.

The Block Circle command duplicates a block in a circle or arc. The Block Circle command can be used to put the teeth on a gear, for example. This command assumes you know the size of the circle and the number of times the block is to be duplicated.

If you have a block you want to duplicate in a circle but don't know the size of the circle, you can use the Polygon command and 3 point circle to find the center of the desired circle. The polygon and circle can be erased after the block is duplicated.

The Block Layer command is useful when you want to change the layer of several lines or entities. The Block Color command can be used to change the color of several lines at a time.

USING GRIDS WITH YOUR DRAWING

DesignCAD has two related but independent grids - the Display Grid and the Snap Grid.

The Display Grid is a grid that is displayed on the screen to aid drawing. The appearance and spacing of the grid can be specified in the Drawing Parameter command.

The Snap Grid is an imaginary grid used to specify the spacing at which points may be set. This can be used, for example, to position all subsequent points in the drawing to be aligned to the nearest .25". Like the Display Grid, the Snap Grid is specified in the Drawing Parameter command (Q key).

The Snap Grid and the Display Grid do not have to be the same size, but it is often useful to make one a multiple of the other. It can also be very helpful to make the Cursor Step Size an even fraction or multiple of the grid sizes.

USING MACROS

The Macro feature of DesignCAD allows you to perform a task one time, record the keystrokes to that task, and repeat it automatically at any time. A macro is actually a program written in the DesignCAD command language.

A Macro can be created, for example, to break a line and insert an object at the break in the line. This can be repeated at different locations by moving the cursor and executing the Macro.

The single keystroke macros (MACRO1 through MACRO10, for keys ALT-F1 through ALT-F10) make it possible to define your own keystroke commands.

When you define a Macro, the keystrokes are record to disk. When you move the cursor one step, it moves a certain distance. When the Macro is later executed, the settings for Cursor Step Size and Snap Grid should remain the same they were when the Macro was created for consistent operation.

The Macro execution begins at the last point set. If no points have been set, the macro begins at the current cursor position. Mouse and digitizer cursor movements cannot be used within a Macro.

PARALLELS, TANGENTS, PERPENDICULARS, AND CIRCLES.

The Parallel command draws a line parallel to another line already in the drawing. The original line can be a straight line, curve, circle, arc, ellipse, or elliptical arc.

If a line is drawn parallel to a curve that has sharp corners, the parallel line may end up with a small loop at the inside of the corner. While this does not look right, it is mathematically correct. This can be prevented by making the parallel line closer to the original, or making the original curve smoother.

A line parallel to an ellipse or an elliptical arc is drawn as another ellipse or elliptical arc moved farther in or out from the center. Because of this, the parallel line is not truly parallel at all points along the ellipse.

If you need a line truly parallel to the ellipse, draw the ellipse using the ALT-O or ALT-A and then do the parallel. A line truly parallel to an ellipse is not an ellipse.

A line can be drawn tangent to any circle or ellipse. The are sometimes two tangent lines that can be drawn to a circle. When this is the case, the tangent that is closer to the second point of the Tangent command will be drawn.

The Perpendicular command draws a line from a given point perpendicular to another line. If you need to draw a perpendicular from a given point on the original line, make a temporary line parallel to the first and draw the perpendicular from the first line to the second.

There are three ways to draw a circle. You can specify the center point and radius, the diameter, or

three points on the outside. If you already know the center point and the size of the radius, you can use the CIRCLE command.

If you know two points between which the circle belongs, use the CIRCLE-2 command. If you have three or more points that the circle lies on, but do not have the center or diameter, then the CIRCLE-3 command can be used.

It is possible to snap to the center of the circle using the Gravity command, making it easy to find the center of the circle after the CIRCLE-2 or CIRCLE-3 command.

AREA AND LENGTH CALCULATIONS

The area and length of a line can be found using the ID command. The Area can be found for a Line, Circle, Ellipse, or Curve.

If a line or curve is not closed, an imaginary line is drawn from the last point to the first point for the area calculation. The length calculation does not require a closed figure, and does not include the line between the first and last points.

The area and length of a polygon can also be found using the AREA command.

The area is calculated in Drawing Units.

SCALING YOUR DRAWING

Your DesignCAD drawing can be output to the printer or plotter at any scale. To do this, set the Drawing Units, and then use the System Parameter command to specify the number of Drawing Units per Inch on Output.

Suppose that you have a house plan drawn in feet, and you want to output that at the scale 1 inch = 4 feet. Set a Drawing Unit to 1 foot in the drawing. Then enter 4 Drawing Units per Inch on Output under the System Parameter command. Next, Enter Y under the option to save the parameters with the drawing.

When the drawing is printed or plotted, it will be printed at the scale of 1 inch = 4 feet, unless the paper is too small to hold the drawing at that scale.

DESIGNCAD UTILITIES

With the DesignCAD file transfer utilities, you can transfer drawing files to and from other programs. There are four translation programs - DCIMPORT, DCEXPORT, DXF, and IGESIN.

DCIMPORT can translate Hewlett Packard plotter files, ASCII text files, and X,Y coordinate files into DesignCAD format. DCEXPORT translates DesignCAD drawing files into IGES (Initial Graphics Exchange Standard), GEM, and Postscript format files.

The DXF Utility transfers drawings to and from DXF file format. IGESIN can input an IGES drawing and translate it into DesignCAD format.

In addition to the file transfer utilities, there are three other DesignCAD utility programs - MATERIAL, BREAKDC, and PLOTSORT. MATERIAL is a materials list program - it lists and counts the attributes in a drawing file.

BREAKDC is used to separate a large DesignCAD drawing into smaller drawing files. This is useful if an extremely large drawing is converted to DesignCAD from another system.

PLOTSORT optimizes a DesignCAD drawing for plotting.

All these programs except the DXF Utility can be found on your DesignCAD Utility Disk. The DXF Interchange program is found on the DesignCAD Configuration disk.

DCIMPORT

The DCIMPORT utility translates files in other formats into DesignCAD drawings. Three types of files can be converted to DesignCAD drawings:

1. HP Plotter Files (HPGL).
2. ASCII Text Files.
3. X,Y Coordinate Files.

With the HPGL conversion, you can read files created on a host of other systems, from large CAD/CAM systems to business graphics packages for personal computers - virtually any program that supports an HP plotter!

The first step in transferring a drawing from another system to DesignCAD is creating a plotter file in HPGL, the HP plotter language. Most programs will give you the option to output the plotter commands to disk. A file created in this manner can then be input to the DCIMPORT utility and translated into a DesignCAD file.

The second input option, ASCII text files, allows a text file created using a word processor to be converted to a DesignCAD drawing. This makes it possible to merge text with your drawings.

The X,Y Coordinate conversion is used to import data points from another program. The input file must contain an X,Y coordinate pair (2 numbers), or an X,Y,Z coordinate triplet (3 numbers) on each line. Each line should be terminated by a carriage return and optionally a line feed character.

The numbers should be separated by one or more spaces. Any number of data points may be used. The data points will be connected with a line or a curve in the resulting DesignCAD drawing.

To use the import utility, type DCIMPORT. (The DCIMPORT utility is found on your DesignCAD disk #2.) You will then be given the following options:

1. Translate an HPGL file to DesignCAD Format.
2. Translate a Text File to DesignCAD Format.
3. Translate a File of X,Y Coordinates to DesignCAD Format.
4. Specify Plotter Size for HPGL File.
5. Stop the Program.

Options 1, 2, and 3 are used to translate files to DesignCAD format. You will be asked for the input file name and the output file name. Enter the file names, including the extension, if any. If you do not specify a file type for the DesignCAD file, .DC2 will be used.

If option 3 is selected, you will be asked for the color, and whether to connect the points with a straight line, a curve, or to place small squares on the points.

Option 4 can be used to select the plotter paper size for which the HPGL file was created. The default size is B size, or 11" by 17". If you are not sure about the size, leave it at the default size. You can alter the size in DesignCAD.

After you are finished, select option 5 to stop the program.

DCEXPORT

The DCEXPORT utility translates DesignCAD drawing files into the the following formats:

1. IGES
2. GEM (Ventura)
3. Postscript

IGES (Initial Graphics Exchange Standard) is a standard format supported by most larger CAD systems. This makes it possible to transfer any DesignCAD drawing to most large CAD systems. GEM and Postscript are graphics file formats used by desktop publishing systems.

To use the DCEXPORT utility, enter DCEXPORT. (The DCEXPORT utility is found on your DesignCAD utility disk.) You will then be given these options:

1. Translate a DesignCAD file to IGES Format.
2. Translate a DesignCAD file to GEM Format.
3. Translate a DesignCAD file to Postscript Format.
4. Stop the program.

To run the program, enter option 1, 2, or 3, according to the file format to be produced. You will be asked for the input file name and the output file name. Enter the file names, including the extension, if any.

The file type or extension .DC2 is assumed for the DesignCAD file, if none is given. A file type of .GEM is used for GEM files, and a file type of .PS is used for Postscript files if none is given.

After you are finished, select option 4 to stop the program.

PLOTSORT

The PLOTSORT utility reads a DesignCAD drawing file, optimizes it for plotting, and writes the file back out to disk. The program sorts the drawing by layer, color, and line location for fast plotting.

This program can be very valuable if you have a drawing that is plotted frequently. Even if the drawing is plotted only once or twice, it may be worthwhile to optimize it with the PLOTSORT program.

To use the program, type PLOTSORT. You will be given three options:

1. Optimize a DesignCAD drawing for plotting.
2. Stop the Program.

Select option 1, and enter the input drawing name. After the drawing has been sorted, enter the output drawing name. This can be the same name as the original drawing.

IGES INPUT PROGRAM

The IGESIN utility provides the capability to read IGES format files produced on other CAD systems, and translate them to DesignCAD format. IGES (Initial Graphics Exchange Standard) is a standard format supported by most larger CAD systems.

To run the program, enter IGESIN. You will be given the following options:

1. TRANSLATE AN IGES FILE TO DESIGNCAD FORMAT.
2. LIST THE DRAWINGS ON DISK.
3. STOP THE PROGRAM.

To transfer an IGES file to DesignCAD format, select option 1 and enter the input and output file names when they are requested.

The following IGES entities are accepted by the IGESIN utility:

100 - Arc
104 - Conic Arc
106 - Copious Data
110 - Line
112 - Parametric Spline
116 - Point
124 - Transformation Matrix
212 - General Note
308 - Subfigure Definition
408 - Subfigure Insertion
410 - View

DXF FILE INTERCHANGE PROGRAM

The DesignCAD - DXF Interchange program provides the capability to transfer DesignCAD drawings to and from thr DXF file format. The DXF file format is compatible with AutoCAD as well as many other software packages.

In addition to allowing you to move files back and forth between DesignCAD and AutoCAD, this makes DesignCAD comapatible with many 3rd party software packages that utilize the DXF format.

To run the program, enter

DXF

You will be asked to select one of the following:

1. TRANSLATE A DESIGNCAD DRAWING TO DXF FORMAT.
2. TRANSLATE A DXF FILE TO DESIGNCAD FORMAT.
3. LIST THE DRAWINGS ON DISK.
4. STOP THE PROGRAM.

To convert a DesignCAD drawing to DXF format, select option 1. To convert a DXF file to DesignCAD format, choose option 2. The program uses a file type of .DC2 for all DesignCAD drawings and .DXF for all DXF files, unless otherwise specified.

After you are finished, select option 4 to stop the program.

MATERIALS LIST PROGRAM

The DesignCAD Materials List Program provides the capability to extract all the Attributes in a drawing, count them, and output the list to the screen, a printer, or a disk file. To use the program, enter

MATERIAL

You will then be asked to select one of the following:

1. LIST THE MATERIALS IN A DRAWING.
2. DISPLAY DESIGNCAD DRAWINGS ON DISK.
3. STOP THE PROGRAM.

To create a materials list for a drawing, select option 1. You will asked for the drawing name, and where you want the output to be directed - screen, printer, or disk.

The screen and printer output is similar in format. The disk output contains the number of occurrances of each attribute and the attribute itself, separated by a comma, on each line.

After you have output the materials list, select option 4 to return to the main menu. You can then produce a materials list for another drawing or exit the program.

ATTRIBUTES

The Materials List program extracts Attributes from a DesignCAD drawing. The Attributes are entered into the drawing using the DesignCAD Attribute command. An attribute is actually a line of text that is associated with a particular position in the drawing.

An attribute can be assigned to a particular object in the drawing, such as a window, door, or electrical component. You can give every "important" object in your drawing an attribute, using identical attributes for identical objects. It is sometimes convenient to put an attribute with a symbol, before it is used, so that when it is retrieved from disk or copied in the drawing the attribute will automatically reproduced.

After the drawing has been completed, with all the attributes assigned, the Materials List program is used to produce a list of all the attributes in the drawing. Remember - the Materials List program only lists and counts attributes. The important part is the correct placement of attributes within your drawing.

USING OTHER SOFTWARE WITH THE MATERIALS LIST PROGRAM

The Materials List program can output the attributes in a drawing to a disk file. This makes is possible to use other software to read the disk file. For example, a database system can get part numbers from the attributes, look up current costs, and produce a cost estimate for the drawing.

The disk file is in the following format:

Columns 1-6:	Quantity, right justified, padded with spaces.
Column 7:	Comma
Column 8+:	Attribute string.

The atrtibutes are listed in alphabetical order. The attribute string is terminated with a line feed and carriage return. The attribute can be no longer than 80 characters.

APPENDIX A: DESIGNCAD DRAWING FILE FORMAT

This section contains advanced information not normally needed by DesignCAD users. Unless you have a specific need for this information, you may skip this section.

The DesignCAD drawing file is an ASCII file, with the data present in character format. Each "record" in the file is actually a line in a text file. There are four types of records, or lines, in the file:

1. Header line
2. Entity line
3. Point line
4. String line

The Header line tells, among other things, how many entity lines are to follow. The entity line tells what type of drawing entity and how many points and strings are to follow. The Point and String lines contain point coordinates and strings, respectively.

THE HEADER LINE

The Header Line contains 5 numbers:

1. The minimum X coordinate in the file.
2. The minimum Y coordinate in the file.
3. The horizontal length of the drawing.
4. The vertical height of the drawing.
5. This number is no longer used. A zero remains here for compatibility.

The coordinates and lengths are given in "dots" on the screen. These values are pertinent only within the drawing file. The drawing may change location, size, etc. when it is retrieved.

THE ENTITY LINE

The entity line contains 5 numbers:

1. The type of entity:
   - 0 = Handles
   - 1 = Line
   - 2 = Oval
   - 3 = Text
   - 4 = Curve
   - 5 = Arc
   - 6 = Paint
   - 15 = Attribute
   - 16 = Circle, Circular Arc
   - 17 = Hatch
   - 18 = Dimension - Linear
   - 19 = Dimension - Angular
   - 20 = System Parameters
   - 21 = New Layer
   - 22 = Text Arc
   - 23 = Layer Names
2. The number of points to follow. (0-200)
   (or the layer number, if entity = 21)
3. The line pattern scale. (1=normal)
   (or the hatch pattern scale)
4. The line width.
5. The line type (0-8)
6. The color (1-16).

Following the entity line are the point lines and the string lines. There is one point per line, and one string per line. Strings are found on Text, Attribute, and Hatch entities.

If the entity = 20, then the drawing parameters will follow, terminated by an asterisk (*).

Earlier versions of the program have the line type and color combined in the same value. This format is

still compatible. If there are only 5 values on the entity line, the color and line type are assumed to be combined on the fifth value.

THE POINT LINE

The point lines follow the entity line. Each line contains two numbers: the X and Y coordinates of the point on the screen. These values will change when the drawing is retrieved according to the size, location, and angle of the Retrieve command.

There can be no more than 200 points per entity. A line or curve entity with more than 200 points may be used by splitting it into two or more entities.

The point line can contain an X, Y, and Z coordinate, but the Z coordinate is ignored.

THE STRING LINE

The string line contains a string of text. It follows the point lines. The string line is used only with the Text, Attribute, and Hatch entities. Be sure that the string is terminated with a carriage return character, because trailing blanks will be used in determining the string length and may cause undesirable results. The maximum string length is 80 characters.

DesignCAD uses the file naming convention of "name.DC2" for drawings. "name" may be any valid MSDOs file name.

SYSTEM PARAMETERS

Following the System Parameter command are the system parameters. These consist of:

Cursor Incerement
Display Grid Type
Display Grid Size
Snap Grid Size
Number of digits accuracy
Default Text Size
Default Text Angle
Units per Inch on Output
Metric (2.54) or English (1) Units (not used)
Fillet Radius
Display Attributes Flag (not used)
Number of X,Y Units per Drawing Unit
Character Font (not used)
Display Grid On or Off
Snap Grid On or Off
Menu On or Off (not used)
Mathematical or Geographical Degrees (not used)
Fill Wide Lines On or Off (not used)
Sound (0, 1, or 2) (not used)
Manipulate other than current layer (not used)
Absolute X, Y offset (not used)
Dimension format
Dynamic dimensioning on or off

Point Definitions:

Entity type 0: 1 or 2 points for the handles of the drawing.

Entity type 1 and 4, Line or Curve - Up to 200 points through which lines are drawn.

Entity type 2 - Oval:

1. Center
2. Radius 1
3. Point through which ellipse passes.
4. Minimum X,Y position of ellipse.
5. Maximum X,Y position of ellipse.

Entity type 3 - Text:

1. Lower left corner of text.
1. Lower right corner of text.
1. Upper left corner of text.
1. Upper right corner of text.

Entity type 5 - Elliptical Arc:

1. Center
2. Beginning Point
3. Ending Point
4. Minimum X,Y position of ellipse. (Optional)
5. Maximum X,Y position of ellipse. (Optional)
6. Radius 1. (Optional)
7. Radius 2. (Optional)

Entity type 15, Attribute - 1 point for location.

Entity type 16 - Circle, Circular Arc:

1. Center
2. Beginning Point
3. Ending Point
4. Minimum X,Y of circle or arc (Optional).
5. Maximum X,Y of circle or arc (Optional).

If points 2 and 3 are the same, a circle is drawn.

Entity type 17 - Hatch - Up to 200 points inside which is to be hatched.

Entity type 18 - Dimension, linear

1. First point for measurement, text.
2. Second point for measurement, text.
3. First point for leader line 1. (optional)

4. Second point for leader line 1. (optional)
5. First point for leader line 2. (optional)
6. Second point for leader line 2. (optional)

Entity type 19 - Dimension, linear
1. Center of angle.
2. First point for "arc".
3. Second point for "arc".
4. First point for leader line 1. (optional)
5. Second point for leader line 1. (optional)
6. First point for leader line 2. (optional)
7. Second point for leader line 2. (optional)

Entity type 22 - Text Arc
1. Center of arc.
2. Beginning point for arc.
3. Ending point for arc.
4. Beginning point for text (= point 2 or 3).

Entity type 23 - Layer Names
One layer name for each "point" designated in the entity line. No points are actually used.

APPENDIX B: FILES USED BY DESIGNCAD

This section contains advanced information not normally needed by DesignCAD users. Unless you have a specific need for this information, you may skip this section.

The following files are used by DesignCAD:

DC.EXE
DCADx.VCT (x = 1 to 9)
DCAD1.SYS
DC2MENU.SYS
DCHATCH.SYS
SETUP.DAT
DCAD2.SYS

DC.EXE is the DesignCAD program. DCADx.VCT is the set of vectors for a DesignCAD character set. Character font 1 is called DCAD1.VCT, 2 is called DCAD2.VCT, etc.

DCAD1.SYS is a file containing the following information:

1. Line type patterns.
2. Digitizing Pad Commands.
3. Printer information.
4. Plotter information.
5. Digitizing pad information.
6. Display information.

Files used to store DesignCAD drawings are called:

name.DC2

where 'name' is the name of the drawing being saved. Files are used to store DesignCAD Macros are called:

name.DCM

where 'name' is the name of the macro being saved. The program SETUP uses data from the file SETUP.DAT pertaining to peripheral devices.

The DesignCAD Menu information is contained in the file DC2MENU.SYS. The Hatching Patterns are found in the file DCHATCH.SYS.

DCAD2.SYS contains the default setup information. This file is created under the Drawing Parameter command or the System Parameter when the values are saved as program default. The file consists of the following items, each on separate lines:

Cursor Incerement
Display Grid Type
Display Grid Size
Snap Grid Size
Number of digits accuracy
Default Text Size
Default Text Angle
Units per Inch on Output
Metric (2.54) or English (1) Units
Fillet Radius
Display Attributes Flag
Number of X,Y Units per Drawing Unit
Character Font
Display Grid On or Off
Snap Grid On or Off
Menu On or Off
Mathematical or Geographical Degrees
Fill Wide Lines On or Off
Sound (0, 1, or 2)
Manipulate other than current layer flag
Absolute X, Y offset (not used)
Save parameters with drawing - on of off
Dimension format
Dynamic dimensioning on or off

APPENDIX C: THE CHARACTER FONT FILES

This section contains advanced information not normally needed by DesignCAD users. Unless you have a specific need for this information, you may skip this section.

DesignCAD uses files named DCADx.VCT to store the vectors used to draw the letters, numbers, and special characters on the screen, where x is the number of the character font. You can modify these files or create your own character sets to be used by DesignCAD.

The file DCADx.VCT is a sequential ASCII file. The characters appear in the following format:

Line 1: The character itself.
Line 2: The number of points used to define the character.
Line 3: The points. (More than one line can be used for the points if necessary.)
Line 4: Character width (in coordinates).

The coordinate range is based on the values at the end of the file.

The points are in the following format:

X1,Y1,...Xi,Yi,C1,C1,X1,Y1,...Xj,Yj,C2,C2,X1,Y1...

X and Y are points used by the command. C is the command:

-1 - vector
-2 - Arc
-3 - Oval
-4 - Curve
-5 - Backwards Arc

The first command is assumed to be a Line command. If no line commands are used, the points may begin with a command.

While a negative Y value is permissible, negative X values should not be used because they could not be differentiated from a command.

At the end of the file, the character is a null line, and three numbers follow on the next line. The first is the length of the X axis defining the character (including the empty space before the next character). The second number is the maximum height of the character, and the third number is the size of the empty space between characters.

A total of 4000 points and 150 characters may be used in a character set file.

APPENDIX D: THE DESIGNCAD SYSTEM FILE

This section contains advanced information not normally needed by DesignCAD users. Unless you have a specific need for this information, you may skip this section.

DesignCAD uses a file named DCAD1.SYS to store the following system information:

1. Line type patterns.
2. Digitizing pad commands.
3. Printer information.
4. Plotter information.
5. Digitizing pad information.
6. Display information.

The file is a sequential ASCII file, logically divided into sections for the above data.

The first two lines of the file contain the Units per Inch, and the name of the units. For example, the inches and centimeters look like this:

1	or	2.54
INCHES		CM

The next line has 4 numbers:

1. Display Grid color.
2. Cursor color.
3. Crosshair color.
4. Points color.

The line type section is located next in the file. On the first line is the number of line type descriptions that follow. Each line type description consists of 11 values on a single line. The first value gives the number of segments in the line pattern. The segment lengths then follow. The segment lengths alternately describe "on" lengths and

"off" lengths. The following is a sample line type section of the file DCAD1.SYS:

```
8
2  6  4  0  0  0  0 0  0  0  0
2  3  2  0  0  0  0 0  0  0  0
4 14  3  3  3  0  0 0  0  0  0
6 14  3  3  3  3  3 0  0  0  0
2  1  3  0  0  0  0 0  0  0  0
4  6  3  1  3  0  0 0  0  0  0
6  6  3  6  3  1  3 0  0  0  0
6  6  3  1  3  1  3 0  0  0  0
```

The first line pattern, for example, consists of a length of 6 "on", followed by a length of 4 "off".

After the line type section is the command section for digitizing pads. The first line of this section tells how many commands can be entered from the digitizing pad. The commands themselves follow, represented by the decimal ASCII values for the command characters. The ASCII code for the ALT keys are the scan code plus 128.

The following is a sample of this section:

```
 48
161  69  33  40  77 123 235 241
 93 122  73  42  68  86 234 240
 66  67  35  61  78  84 233 239
 64  43  71  80  45 162 232 238
 62  72  70  79  91  83   0 237
 65  89  27 175  34  88  90 236
 8 7 40 8 2.33333
 0 126 125 86 67 79 70 161 86 78 34 42 68 69 27 33
 0 126 125 67 86 27 161
```

For more information on this section, See Appendix I.

The next section of the file contains the printer information. The printer section is organized in the following format:

1. Printer Name
2. Printer Type:
   - 0 = Epson Type Format
   - 1 = Toshiba Format
   - 2 = Okidata, IDS Format
   - 4 = NEC Format
   - 5 = LQ-1500 Format
   - 6 = HP Laser Jet
   - 7 = Anadex
   - 8 = MPi
   - 9 = CGP-220
   - 10 = DMP-2100
   - 11 = Printek
   - 12 = Printronix
   - 13 = HP Laser Jet Plus
   - 14 = IBM Quietwriter II
   - 15 = IBM Colorjet
   - 16 = Xerox 4045
   - 17 = Raster
   - 18 = Okidata 293
   - 19 = C. Itoh 24 LQ
3. A. Output Device used by the printer:
   - 0 - LPT1:
   - 1 - LPT2:
   - 2 - COM1:
   - 3 - COM2:
   - 4 - Disk File

   B. Baud Rate

0 - 110	4 - 1200
1 - 150	5 - 2400
2 - 300	6 - 4800
3 - 600	7 - 9600
	8 - 19200

   C. Parity
   - 0 - No Parity
   - 1 - Odd Parity
   - 3 - Even Parity

D. Number of Stop Bits
E. Word Length
Item B, C, D, and E must be present, although they won't be used, if the printer device is the parallel port.

4. Number of densities of graphics (1 to 3).
   For Each Density:
   5. Number of characters in graphics control code.
   6. Control codes for graphics mode.
   7. Count type:
      0 = Two byte binary
      1 = Four byte ASCII
      2 = No count field
   8. Interleave factor - 0, 1, or 2.
      For each phase: (Interleave factor+1)
      9. Number of characters in line feed control code.
      10. Line feed control codes.
   11. Horizontal dots per inch.
   12. Vertical dots per inch.
      13. Number of characters in end control code.
      14. Ending control codes.
      15. Maximum usable paper width.
16. Number of Color Planes (Ribbon Color Bands).
   For each Color Plane:
   17. Number of characters in color control code.
   18. Control codes for color band change.
(If 16 is non-zero, then 19 and 20 are present).
19. Number of Colors.
20. Color Masks - one for each color.
    The bits of a color mask tell which color planes (ribbon color bands) make up that color.

Following the printer information is the plotter information. The following format is used for this section:

1. Plotter Name
2. 0 = Normal, 1 = Houston Instruments
3. A. Output Device used by the plotter:

0 - LPT1:
1 - LPT2:
2 - COM1:
3 - COM2:
4 - Disk File

B. Baud Rate

0 - 110	4 - 1200
1 - 150	5 - 2400
2 - 300	6 - 4800
3 - 600	7 - 9600
	8 - 19200

C. Parity
0 - No Parity
1 - Odd Parity
3 - Even Parity

D. Number of Stop Bits

E. Word Length

Item B, C, D, and E must be present, although they won't be used, if the plotter device is the parallel port.

4. Separator character (between numbers).
5. Terminator character (ends commands).
6. Number of Pens (0 = Pause to Change).
7. X-Min, X-Max, Y-Min, Y-Max.
8. Move Absolute command.
9. Draw Absolute command.
10. Initialization command (Before each drawing).
11. Change Pens command.
12. Command After Plotting is Complete.
13. Movement increments per inch.

Following the plotter information is the digitizing tablet information. The following format is used for this section:

1. Tablet name
2. Tablet attribute byte:
   (Values for ASCII Digitizers Only:)
   Bit 0 - 0=No Echo, 1=Echo
   Bit 1 - 0=Trigger Mode, 1=Stream
   Bit 2 - 0=Button Last, 1=Button First
   Bit 3 - 0=Ignore Space, 1=Space Separator
   Bit 4 - 0=No Sign, 1=Sign Precedes Point
   Bit 5 - 0=Normal, 1=Hi Pad format.
   Bit 6 - 0=Normal, 1=Summa Binary format.
   Bit 7 - 0=Normal, 1=Unique Format
   Tablet ID (Arbitrary Value)
3. Tablet width, in inches.
   Tablet height, in inches.
4. X value at lower left corner.
   Y value at lower left corner.
   Tablet resolution in lines per inch.
5. Length of trigger string
6. Trigger string ASCII decimal values. (65=A)
7. Length of init string
8. Initialization string ASCII decimal values.
9. Serial Port (0=COM1:, 1=COM2:)
   Baud - 7 = 9600
          6 = 4800
          5 = 2400
          4 = 1200
          3 = 600
          2 = 300
          1 = 150
          0 = 110
   Parity - 0 = None
            2 = Even
            3 = Odd
   Stop Bits - 0 or 1
   Word Length - 7 or 8

Following the digitizing tablet information is the display information. The following format is used for this section:

1. Display Name
2. Display Type
   Minimum X on drawing area.
   Minimum Y on drawing area.
   Maximum X on drawing area.
   Maximum Y on drawing area.
   Number of non-black colors.
   Text mode BIOS value.
   Graphics mode BIOS value.
   Number of buffer pages required.
3. Aspect ratio.
4. Color masks - 16 values.

The color map is a set of sixteen values that correspond to colors one through 16 in DesignCAD. If you need to modify the color selections, you can edit the file and change these values.

APPENDIX E: PROGRAM LIMITATIONS

1. The maximum number of points set at a time is 200.
2. The maximum number of points used in a drawing is 10,000 to 32,000, depending on available memory.
3. The maximum number of entities used in a drawing is 2,000 to 10,000, depending on available memory.
4. The maximum number of text strings used in a drawing is 2000.
5. The maximum number of vectors in the file DCADx.VCT is 4000.
6. The maximum characters in the file DCADx.VCT is 150.

APPENDIX F: DEFINING YOUR OWN PLOTTER

The program SETUP is used to choose the type of plotter you use. If your particular plotter is not listed as an option, you can define your device for use with DesignCAD.

DEFINING A PLOTTER

To setup DesignCAD for a custom plotter, select the option "A PLOTTER OTHER THAN THESE" when asked which plotter you use. Next, enter the information that is displayed on the screen:

1. PLOTTER NAME

2. OUTPUT DEVICE
   This is the logical device to which your plotter is connected:
   0 - LPT1:
   1 - LPT2:
   2 - COM1:
   3 - COM2:.

3. SEPARATOR
   The separator is the character which is output to the plotter between points. It is usually a comma or a space.

4. TERMINATOR
   The terminator is the character that comes at the end of each plotter command. (DesignCAD automatically outputs a Carriage Return and Line Feed after each command.) Leave this blank if none is required.

5. NUMBER OF PENS
   Enter the number of pens your plotter uses

automatically. If a manual pen change is required, enter 0. If only 1 pen is to be used regardless of color, enter 1.

6. X MINIMUM
   This is the minimum X coordinate for your plotter.

7. X MAXIMUM
   This is the maximum X coordinate for your plotter.

8. Y MINIMUM
   This is the minimum Y coordinate for your plotter.

9. Y MAXIMUM
   This is the maximum Y coordinate for your plotter.

10. MOVE COMMAND
    Enter the plotter command used to "Move Absolute", or to move from the current location to the point specified with the pen up.

11. DRAW COMMAND
    Enter the plotter command used to "Draw Absolute", or to draw from the current location to the points specified with the pen down.

12. SETUP COMMAND
    If an initialization command is required for your plotter, enter it here. It will be output to the plotter before each drawing is plotted.

13. PEN CHANGE COMMAND
    If your plotter has more than one pen,

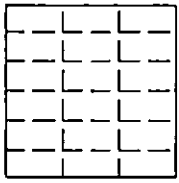
enter the command used to change from one pen to another. The number of the new pen will be provided by DesignCAD.

14. MOTION INCREMENTS PER INCH
    This value is the number of motion increments per inch. The number can be calculated by dividing the pen range by the maximum value.

These items are all listed on the screen. You may use the arrow keys to move from one to another. If you want to return to the previous option screen, use the ESC key.

Another way to define a plotter is to choose a plotter in the SETUP program and then edit the file DCAD1.SYS. In the system file, the name of the plotter is followed by the plotter parameters. The parameters can be changed with a text editor. For the format of this file, see Appendix D.

DESIGNCAD HATCHING PATTERNS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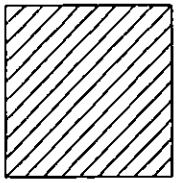
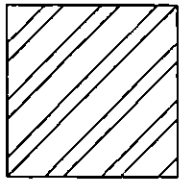

ANSI31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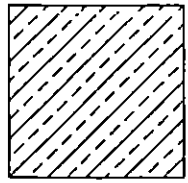
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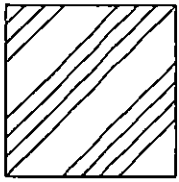
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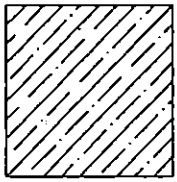
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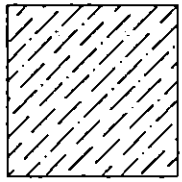
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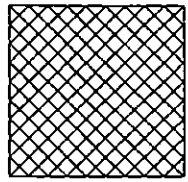
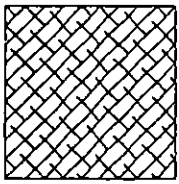
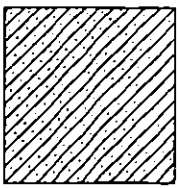

ANSI37


ANSI38

BRASS

BRICK

CLAY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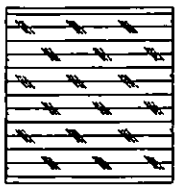
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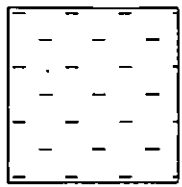
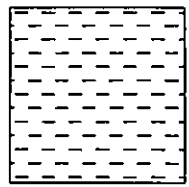
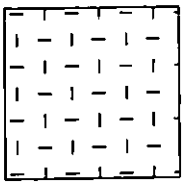

CROSS


DASH

DESIGNCAD HATCHING PATTERNS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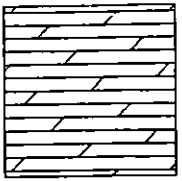
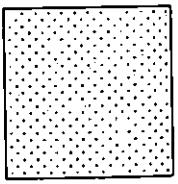
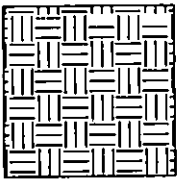

DOLOMITE


DOTS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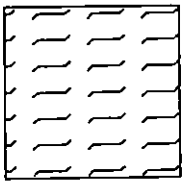
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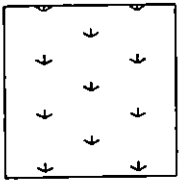
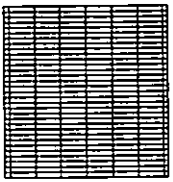

GRASS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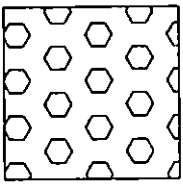
HEATRANS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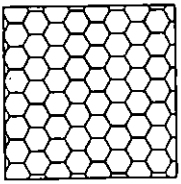
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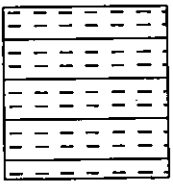
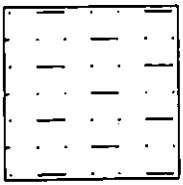
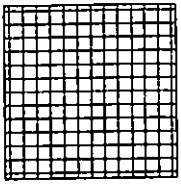
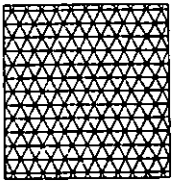

INSULATE

LINES


MUD


NET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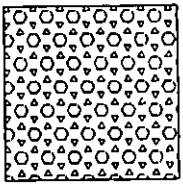
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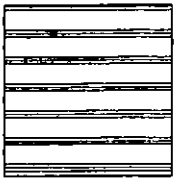
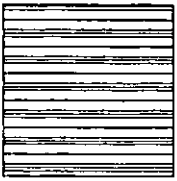
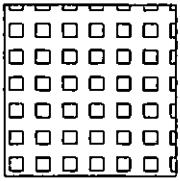
DESIGNCAD HATCHING PATTERNS



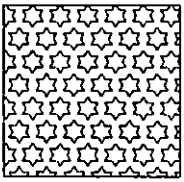
PLASTIC



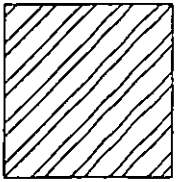
PLASTIC1



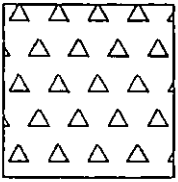
SQUARES



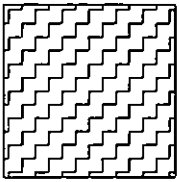
STARS



STEEL



TRIANGLE



ZIGZAG

APPENDIX H - CUSTOMIZING THE COMMAND MENU

This section contains advanced information not normally needed by DesignCAD users. Unless you have a specific need for this information, you may skip this section.

The Command Menu is the list of commands down the right side of the screen. It can be enabled or disabled using the System Parameter.

The Command Menu can be modified to suit your particular needs by editing the menu file "DC2MENU.SYS" with a text editor or a word processor in ASCII mode. The format of the file is:

```
Number of Menus
Separater Line (can be anything - not used)
  Menu 1
  Separater Line
  Menu 2
  Separater Line
  etc.
```

Each Menu has the format:

```
Number of Selections
Menu Selection 1
Menu Value 1
Menu Selection 2
Menu Value 2
Menu Selection 3
Menu Value 3
etc.
```

The Menu Selection can be one of two things:

1. The actual DesignCAD Menu command, or
2. A name for another menu.

If the Menu Selection is a command, then the Menu Value is zero. If the Menu Selection is the name for

another menu, then the Menu Value is the number of that menu. (Menus are numbered in the order that they appear in the file.) The Menu Selection can be a maximum of 9 characters long.

A blank Menu Selection or a Menu Selection ending with a colon (for example, ROOTMENU:) cannot be selected. The entries are used to make the menu more readable.

Here is a sample menu file containing two menus:

```
2
================  <This line is just for looks>
6                 <6 entries in this menu>
MENUNAME:         <Selections ending in colon are
0                  not executed>
                  <Blank selections are not
0                  executed>
ARC               <The zero 0 means that this is
0                  a command>
ARC-2
0

0
NEXT MENU         <The 2 means go to menu 2 when
2                  this selection is chosen.>
================
7
MENU 2:
0

0
CURVE
4
TEXT
0
ARROW
0

0
```

```
LAST MENU
1
```

The two menus will look like this on the screen:

```
Menu 1:                    Menu 2:
-------------              ---------------
  MENUNAME:                  MENU 2:

  ARC                        CURVE
  ARC-2                      TEXT
                             ARROW
  NEXTMENU
                             LAST MENU
```

APPENDIX I - CUSTOMIZING DIGITIZER MENU

This section contains advanced information not normally needed by DesignCAD users. Unless you have a specific need for this information, you may skip this section.

The Commands for the Digitizer menu are located in the file DCSETUP.DAT. You can design your own digitizer menu by editing this file with a text editor or a word processor in ASCII mode. Before you modify this file, be sure you have another copy of it in case you make mistakes.

The Digitizer Menu can be modified to suit your particular needs by editing the file "DCSETUP.DAT". The first part of the file looks like this:

```
19
1
INCHES
8
2 21 12  0  0  0  0  0  0  0  0
4 12  6 12  6  0  0  0  0  0  0
4 21  6  3  6  0  0  0  0  0  0
6 14  6  3  6  3  6  0  0  0  0
6  1 12  1 12  1 12  0  0  0  0
4 23  6  1  6  0  0  0  0  0  0
6  9  6  9  6  1  6  0  0  0  0
6 18  6  1  6  1  6  0  0  0  0
48
161  69  33  40  77 123 235 241
93  122  73  42  68  86 234 240
66   67  35  61  78  84 233 239
64   43  71  80  45 162 232 238
62   72  70  79  91  83   0 237
65   89  27 175  34  88  90 236
8 7 40 8 2.33333
0 126 125 86 67 79 70 161 86 78 34 42 68 69 27 33
0 126 125 67 86 27 161
=======================================
```

The first number in the file tells how many lines there are before the line of equal signs. The information for the digitizer menu starts on the 13th line with the number 48. 48 tells how many digitizer commands there are.

Then come the commands themselves, organized by row beginning at the bottom row of the digitizer menu. All the digitizer commands are keystroke commands, represented by the ASCII code for the keystroke character. For example, the number 65 represents the keystroke command A (Arc).

After the commands come the following 5 numbers:

a) 8 - eight commands per row.
b) 7 - 7 rows of commands, including the alphabet.
c) 40 - 40 character across the top.
d) 8 - digitizer menu is 8 inches wide.
d) 2.33 - digitizer menu 2.33 inches tall.

The next two lines are the ASCII values for the button commands on a digitizing tablet, and for a mouse. These can be modified to change the command sequence for the buttons on the digitizing tablet or mouse. There are sixteen values for the digitizer and six for the mouse, although they may not all be used.

The top row of the digitizer menu is a set of characters that can be selected with the digitizer cursor or stylus. These characters, as well as the number 40, should not be changed.

Any of the other numbers can be changed to suit your digitizer menu. The maximum number of commands on a digitizer menu is 255.

After you have modified the file DCSETUP.DAT, you must run the DesignCAD Configuration program SETUP

and save the configuration information before you can use the new digitizer menu.

Remember to change the first number in DCSETUP.DAT if you change the number of lines before the first set of equal signs (=).

APPENDIX J - DEFINING YOUR OWN HATCH PATTERNS

This section contains advanced information not normally needed by DesignCAD users. Unless you have a specific need for this information, you may skip this section.

The Hatch and Hatch Line commands use hatch patterns found in the file DCHATCH.SYS. You can use your own hatch patterns with DesignCAD by adding them to this file. The file DCHATCH.SYS is an ASCII file that can be edited with a text editor or a word processor in ASCII mode. Be sure you have a second copy of the file DCHATCH.SYS before you modify it.

The file DCHATCH.SYS consists of a series of hatch patterns. Each hatch pattern is in the following format:

1. Hatch Pattern Name
2. Number of Line Definitions
3. Line Definitions, each consisting of
   A. Angle
   B. X offset of first occurence
   C. Y offset of first occurence
   D. X change from first to second occurance.
   E. Y change from first to second occurance.
   F. six numbers defining the line pattern:
      a) length on
      b) length off
      c) length on
      d) length off
      e) length on
      f) length off
      These values are similar to the line type definitions in the file DCAD1.SYS.

For example, the hatch pattern definition for a brick pattern is:

BRICK

```
3
0   0  0  0 20  40   0 0 0 0 0
90  0  0  0 40  20 20 0 0 0 0
90 20 20  0 40  20 20 0 0 0 0
```

There are three lines in the pattern. The first line is a solid line because there is only one non-zero value in the last 6 numbers. It is oriented at an angle of zero, so it is a horizontal line. The first location is at 0,0 (a relative position). For each subsequent location, the X (horizontal) location remains the same (which is irrelevent for a solid line), and the Y location is incremented by 40.

The next line is a vertical line, oriented at 90 degrees. It is a dashed line, with a length of 20 on and 20 off. The first location is at 0,0, and each subsequent line is located at an offset of 40 in the Y direction. The offset of 40 in the Y direction is actually a horizontal offset because of the 90 degree orientation. The offsets are taken as if the line were horizontal.

The third line is the same as the second, but the location of the first occurrence is at 20,20.

A hatch pattern equivalent to the above example is:

```
BRICK
2
0   0  0  0 20  40   0 0 0 0 0
90  0  0 20 20  20 20 0 0 0 0
```

Here, the second and third lines are combined with an offset of 20,20 rather than 0,40.

The line pattern lengths are relative lengths, which are affected by the scale entered by the hatch command.

APPENDIX K - HARDWARE INTERFACING

DIGITIZER - CALCOMP 2100

0 = Down (Open)
1 = Up (Closed)

Switch 1

1	1	8 Bits
2	0	Parity Enabled
3	1	Even Parity
4	0	Disable Fixed Parity
5	0	Disable Parity Check
6	1	9600 Baud
7	0	"
8	0	"

1 2 3 4 5 6 7 8
1 0 1 0 0 1 0 0

Switch 2

1	0	English Scaling
2	0	ASCII Data
3	0	LF After CR
4	0	Remote Mode
5	0	"
6	1	"
7	1	"
8	0	"

1 2 3 4 5 6 7 8 9 10
0 0 0 0 0 1 1 0 X X X = Don't Care

Two jumpers beside switches should be horizontal to avoid having to use a null modem.

APPENDIX K - HARDWARE INTERFACING

DIGITIZER - CALCOMP 2500

BANK	ASCII	BINARY	9100
1	00000001	00000001	00000001
2	11000100	11000100	11000100
3	00000000	00011100	01100100
4	00100001	00100001	00100001
5	01111100	01111100	01111100
6	00000000	00000000	00000000

BINARY IS FASTER.

APPENDIX K - HARDWARE INTERFACING

DIGITIZER - CALCOMP 9100 DIGITIZER

AREA 1, SB1:

```
  0 RUN MODE
  0 RUN MODE
  1 FORMAT 1
  1 FORMAT 1
  1 1000 LINES PER INCH
  1 1000 LINES PER INCH
  X (TABLET SIZE)
  X (TABLET SIZE)
```

AREA 2, SB1:

```
  1 EVEN PARITY
  0 EVEN PARITY
  0 EVEN PARITY
  1 ONE STOP BIT
  1 SEVEN DATA BITS
  1 9600 BAUD
  1 9600 BAUD
  0 9600 BAUD
```

AREA 2, SB2: USE FACTORY SETTINGS
AREA 3, SB2: USE FACTORY SETTINGS

APPENDIX K - HARDWARE INTERFACING

DIGITIZER - GTCO SERIES 5

```
0 0 1 1 0 0 0 1     Switch 1
1 1 1 1 1 0 0 0     Switch 2
0 0 1 1 x 0 1 0     Switch 3
```

Uses Packed Binary High Resolution. Must have 8 bit word.

Switch 1

1 Baud Setting (9600 = 0 0 1 1)
2 "
3 "
4 "
5 Parity Enable
6 Parity Even
7 2 Stop Bits
8 8 Bit Word

Switch 2

1 Include Pushbutton Code (ASCII)
2 Include Space (ASCII)
3 Include CR (ASCII)
4 Include LF (ASCII)
5 Hi Resolution
6 Disable Serial Port B
7 Disable Serial Port A
8 Enable Audible Alarm

Switch 3

1 GPIB instead of RS-232
2 Output Rate
(0 0 =25, 1 0 = 100, 0 1 = 200, 1 1 = Incremental)
3 "
4 Continuous Mode (Stream)
5 16 Button Cursor
6 Metric Data
7 Packed Binary Format
8 Serial Handshaking? Leave Off

There are two serial ports. One requires a null modem, and one doesn't.

APPENDIX K - HARDWARE INTERFACING

DIGITIZER - HITACHI TIGER

(ASCII)
Stream Mode, High Resolution (1000 DPI)

Switch 1: 1 0 0 0 0 0 0 0
Switch 2: 0 0 1 0 1 0 0 0
Switch 3: 0 0 0 0 0 0 0 0

(Even Parity)

(BINARY)

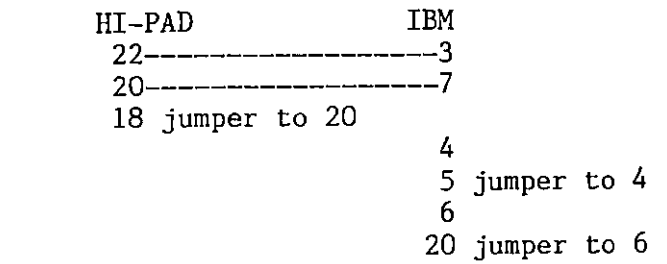
Switch 1: 1 0 0 1 0 0 0 1
Switch 2: 0 0 1 0 1 0 0 0
Switch 3: 1 0 0 0 0 0 1 0

Stream Mode, Low Resolution
(Use Kurta Series 2 Setup for High Resolution)

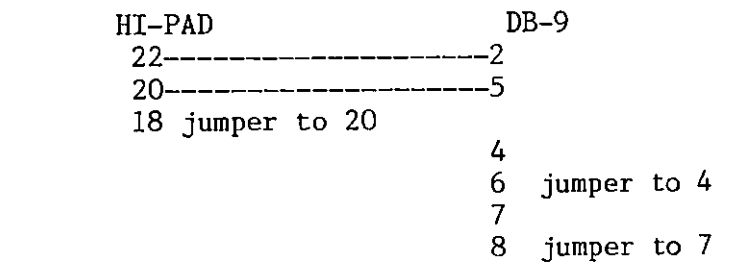
APPENDIX K - HARDWARE INTERFACING

DIGITIZER - HOUSTON INSTRUMENTS DIGITIZERS

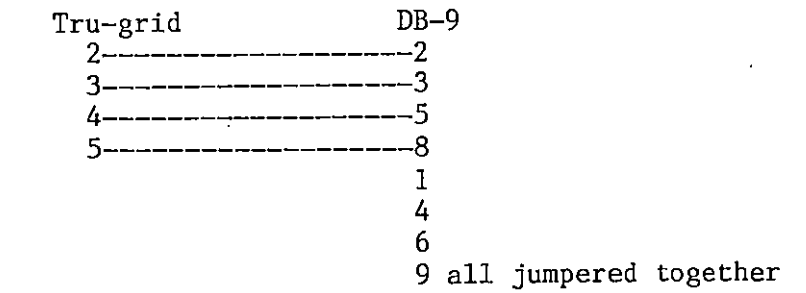
HI-PAD Cable



HI-PAD Cable to DB-9



TRUE-GRID Cable



APPENDIX K - HARDWARE INTERFACING

DIGITIZER - KURTA SERIES THREE-

BANK OF 8 - ALL OFF	00000000
BANK OF 4 - 1 ON, 2-3-4 OFF	1000

DIGITIZER - KURTA SERIES ONE-
(TANDY GT-2000)

BANK OF 8 - 1-7 OFF, 8 ON	00000001
BANK OF 4 - 1 ON, 2-3-4 OFF	0001

PIN 4 MUST BE DISCONNECTED ON THE COMPUTER SIDE OF THE CABLE.

APPENDIX K - HARDWARE INTERFACING

DIGITIZER - MITSUBISHI GRAFNET 01

SHOULD BE 1001000000

	OFF	ON
1	ASCII	BINARY
2	OUTPUT MODE (ON-ON = STREAM)	
3	OUTPUT MODE (ON-ON = STREAM)	
4	INCH	MILLIMETER
5	BAUD (ON-ON = 9600)	
6	BAUD (ON-ON = 9600)	
7	PARITY (ON-ON = EVEN)	(OFF-ON = NONE)
8	PARITY (ON-ON = EVEN)	(OFF-ON = NONE)
9	OPERATING SPEED (ON-ON = 60 PPS)	
10	OPERATING SPEED (ON-ON = 60 PPS)	

STRAIGHT THROUGH CABLE.

APPENDIX K - HARDWARE INTERFACING

DIGITIZER - NUMONICS 2200 DIGITIZER

0 = Down (Open)
1 = Up (Closed)

A-1	1	No Point Mode
A-2	0	Stream Mode
A-3	1	No Incremental Mode
A-4	1	No Switch Stream Mode
A-5	0	English Units
A-6	0	Absolute Mode (*)
A-7	0	Packed Binary
A-8	x	Not Used (*)

(* = 1 for Calcomp 2000 Version)

B-1	1	No CR
B-2	1	No LF
B-3	1	No Parity
B-4	1	Even Parity
B-5	1	1 Stop Bit
B-6	1	No Audible
B-7	1	No XON/XOFF
B-8	1	No Self Diagnostics

APPENDIX K - HARDWARE INTERFACING

DIGITIZER - SAC GP-7, GP-8 DIGITIZER

SWITCH 1:
0 1 1 0 1 0 0 1 0 1

STRAIGHT THROUGH CABLE

MUST BE PLACED IN STREAM MODE MANUALLY
SCRIPTEL DIGITIZER

SWITCH 1:
1 1 0 1 1 0 1 1 0

SWITCH 2:
1 0 0 0 0 0 0 0 0 (ASCII)
1 1 0 0 0 0 0 0 0 (BINARY)

STRAIGHT THROUGH CABLE

APPENDIX K - HARDWARE INTERFACING

DIGITIZER - SUMMAGRAPHICS MICROGRID

		ASCII	BINARY
SW 1 -	1	1	1
	2	0	0
	3	0	0
	4	1	1
	5	1	1
	6	0	0
	7	0	0
	8	0	0
SW 2 -	1	0	0
	2	1	1
	3	0	0
	4	0	1
	5	1	1
	6	1	1
	7	1	1
	8	1	1

ASCII IS FACTORY SETTING, BINARY IS EXCEPT 2-4.
FACTORY SETTING IS 9600, E, 7, 2

THERE ARE TWO RS-232 JACKS. USE THE FIRST ONE WITH A NULL MODEM.

APPENDIX K - HARDWARE INTERFACING

DIGITIZER - SUMMA BIT PAD 2 -

SWITCH 1		
REMOTE ENABLE - OFF	0	
PROXIMITY TRANSMIT	0	
STREAM MODE	1	
SWITCH MODE	0	
ABSOLUTE MODE	0?	
REPORT RATE - 100	1	
REPORT RATE - 100	1	
REPORT RATE - 100	0	
SWITCH 2		
ASCII/BINARY	0	BINARY
CRLF TERMINATOR	1	
INCREMENT SETTING	0	0
INCREMENT SETTING	0	
INCREMENT SETTING	0	
RESOLUTION SETTING	0	200
RESOLUTION SETTING	1	
RESOLUTION SETTING	0	
SWITCH 3		
PARITY ENABLE	0	OFF
EVEN/ODD	0	EVEN
STOP BITS	0	1
CTS HANDSHAKE	1	ENABLE?
CURSOR OUTPUT CODE:	0	B
BAUD	1	9600
BAUD	1	
BAUD	0	

APPENDIX K - HARDWARE INTERFACING

PLOTTER - HEWLETT PACKARD 7475

SWITCH 1:
0 0 0 1 0 1 0 1 0

Use HP Cable 17255 or equivalent
XON/XOFF may be either enabled or disabled.
Use Stand Alone mode, not eavesdrop.

PLOTTER - HEWLETT PACKARD 7586

The following settings are necessary:

Full Duplex
Hardware Handshaking
DTR Normal
XON/XOFF may be either enabled or disabled.
Use Stand Alone mode, not eavesdrop.

Use HP Cable 17255 or equivalent

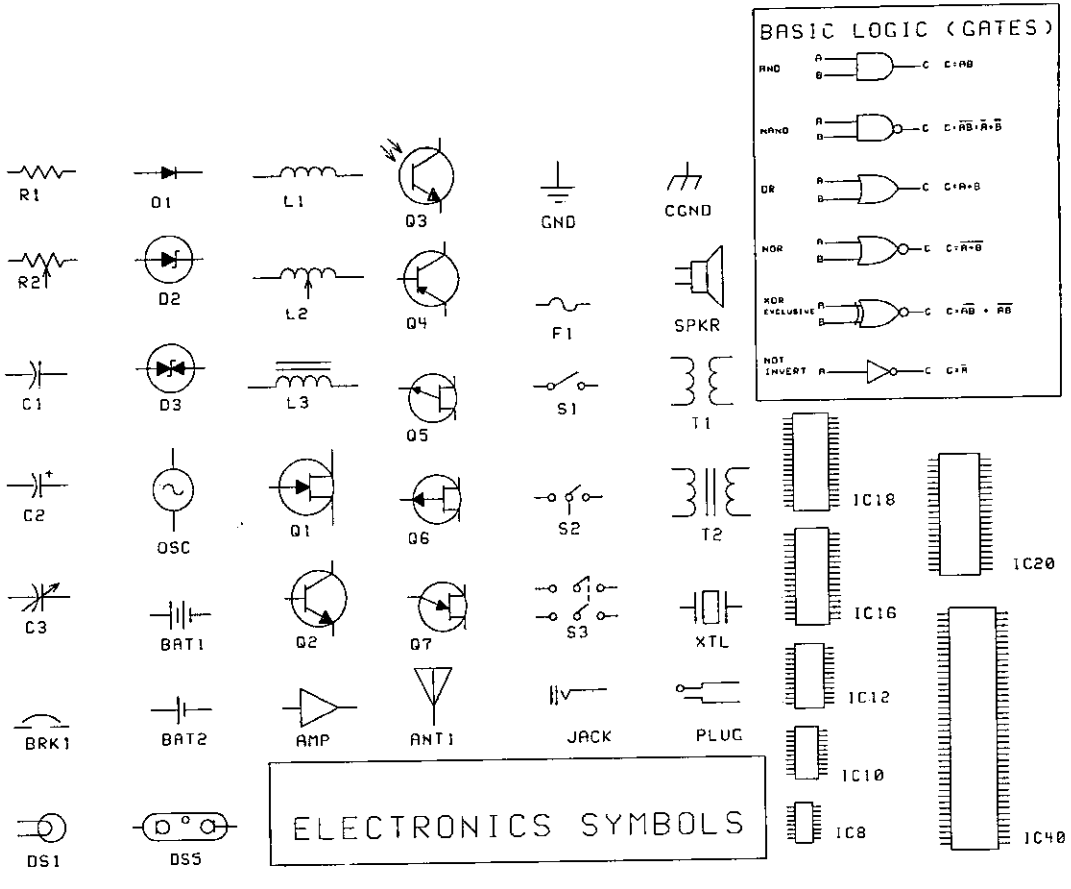
APPENDIX K - HARDWARE INTERFACING

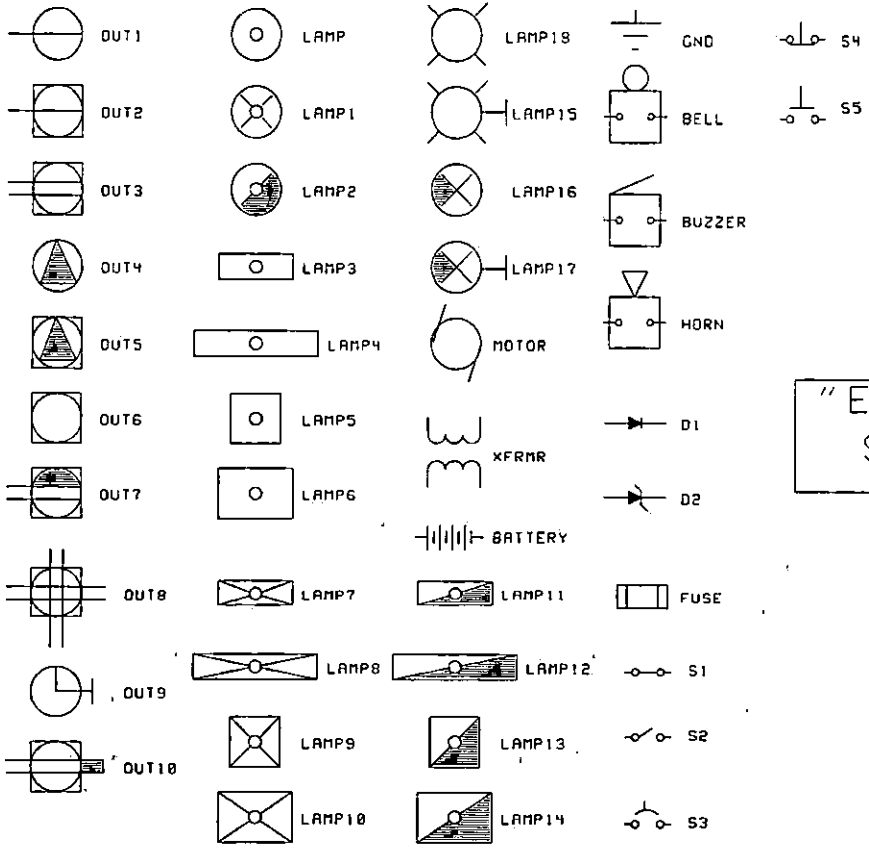
DISPLAY - AT&T 6300

The AT&T 6300 display is available in monochrome, color, and color with the Display Enhancement Board. The standard color version is not capable of 640x400 16 color graphics. It should be set up either as the AT&T 6300 monochrome or the IBM color graphics adapter.

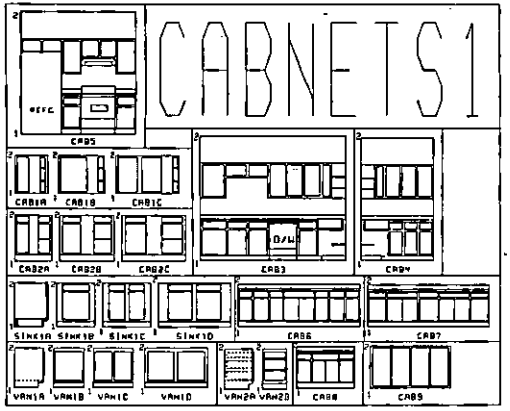
DISPLAY - TOSHIBA 1100

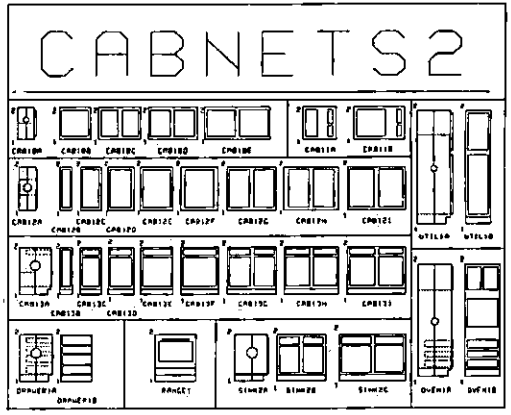
The Toshiba 1100 should be configured as an IBM color graphics adapter with a monochrome display.

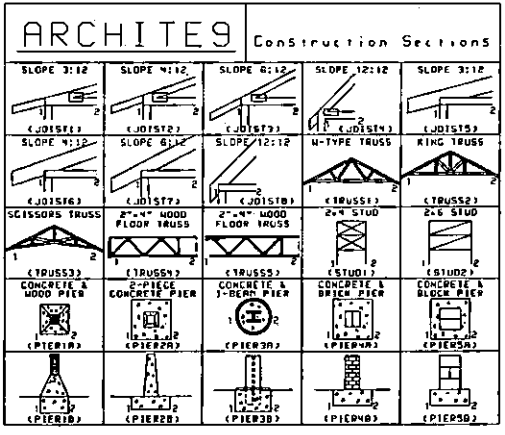

BASIC LOGIC (GATES)
AND
NAND
OR
NOR
XOR EXCLUSIVE
NOT INVERT
R1
D1
L1
Q3
GND
CGND
R2
D2
L2
Q4
F1
SPKR
C1
D3
L3
Q5
S1
T1
C2
OSC
Q1
Q6
S2
T2
IC18
IC20
C3
BAT1
Q2
Q7
S3
XTL
IC16
BRK1
BAT2
AMP
ANT1
JACK
PLUG
IC12
IC10
IC8
IC40
DS1
DS5
ELECTRONICS SYMBOLS

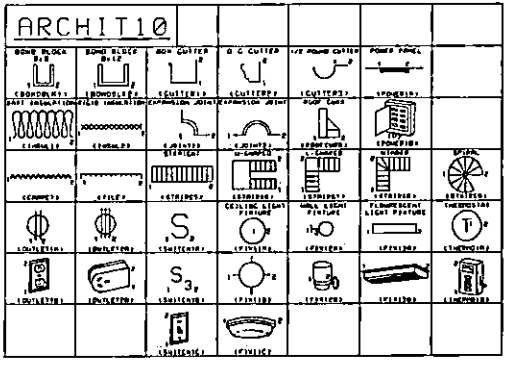


"ELECTRIC" SYMBOLS


CABNETS1
REFG
CAB5
CAB1A
CAB1B
CAB1C
CAB2A
CAB2B
CAB2C
D/W
CAB3
CAB4
SINK1A
SINK1B
SINK1C
SINK1D
CAB6
CAB7
VAN1A
VAN1B
VAN1C
VAN1D
VAN2A
VAN2B
CAB8
CAB9


CABNETS2
CAB10A
CAB10B
CAB10C
CAB10D
CAB10E
CAB11A
CAB11B
CAB12A
CAB12B
CAB12C
CAB12F
CAB12G
CAB12H
CAB12I
CAB12E
CAB12D
UTIL1A
UTIL1B
CAB13A
CAB13C
CAB13E
CAB13F
CAB13G
CAB13H
CAB13I
CAB13D
DRAWER1A
DRAWER1B
RANGE1
SINK2A
SINK2B
SINK2C
OVEN1A
OVEN1B


ARCHITE9
ConStruction Sections
SLOPE 3:12
SLOPE 4:12
SLOPE 6:12
SLOPE 12:12
SLOPE 3:12
(JOIST1)
(JOIST2)
(JOIST3)
(JOIST4)
(JOIST5)
SLOPE 4:12
SLOPE 6:12
SLOPE 12:12
W-TYPE TRUSS
KING TRUSS
(JOIST6)
(JOIST7)
(JOIST8)
(TRUSS1)
(TRUSS2)
SCISSORS TRUSS
2"x4" WOOD FLOOR TRUSS
2"x4" WOOD FLOOR TRUSS
2x4 STUD
2x6 STUD
(TRUSS3)
(TRUSS4)
(TRUSS5)
(STUD1)
(STUD2)
CONCRETE & WOOD PIER
2-PIECE CONCRETE PIER
CONCRETE & I-BEAM PIER
CONCRETE & BRICK PIER
CONCRETE & BLOCK PIER
(PIER1A)
(PIER2A)
(PIER3A)
(PIER4A)
(PIER5A)
(PIER1B)
(PIER2B)
(PIER3B)
(PIER4B)
(PIER5B)

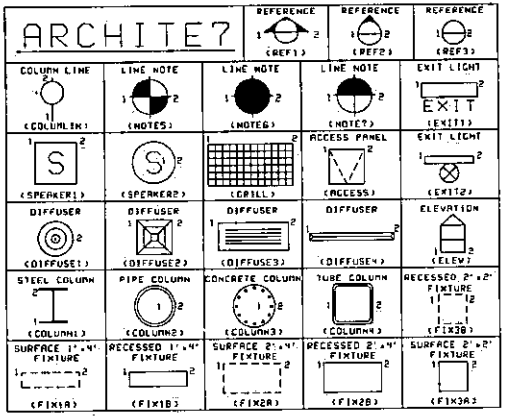

ARCHIT10
BOND BLOCK 8x8
BOND BLOCK 8x12
90° GUTTER
D.C. GUTTER
1/2 ROUND GUTTER
POWER PANEL
(BOND8X8)
(BOND8X12)
(GUTTER1)
(GUTTER2)
(GUTTER3)
(PNL1)
PART INSULATION
FULL INSULATION
EXPANSION JOINT
EXPANSION JOINT
ROOF CURB
(INSUL1)
(INSUL2)
(JOINT1)
(JOINT2)
(CURB1)
(CURB2)
(STAIRS1)
(STAIRS2)
(STAIRS3)
(STAIRS4)
(STAIRS5)
(STAIRS6)
(STAIRS7)
(STAIRS8)
(STAIRS9)
(OUTLET1A)
(OUTLET1B)
(SWITCH1A)
CEILING LIGHT FIXTURE
WALL LIGHT FIXTURE
FLUORESCENT LIGHT FIXTURE
THERMOSTAT
(OUTLET2A)
(OUTLET2B)
(SWITCH2A)
(FIX1A)
(FIX1B)
(FIX1C)
(FIX1D)
(THERMO1A)
(SWITCH2B)
(FIX1E)

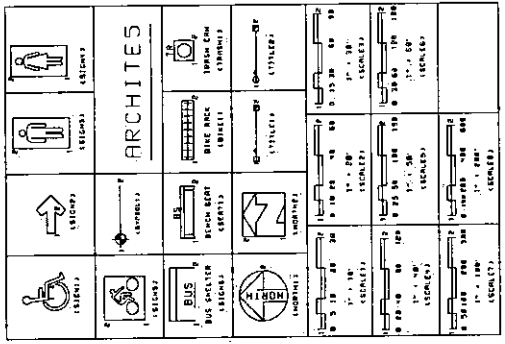
ARCHITE1
WINDOWS
DOORS
STAIRS
CLOSETS
BATH FIXTURES
APPLIANCES
ASSORTED SYMBOLS

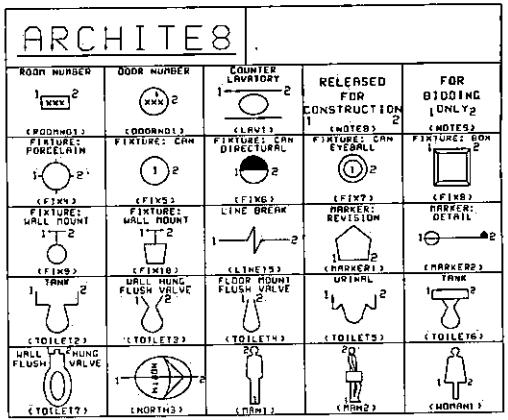
ARCHITE3
SEWER LINE
STORM LINE
GAS LINE
WATER LINE
TELEPHONE LINE
ELECTRIC LINE
PROPERTY LINE
SETBACK LINE
CENTER LINE
EASEMENT LINE
LIGHT POLE
POWER POLE
TELEPHONE POLE
FIRE HYDRANT
FIRE LINE CONNECTOR
GAS VALVE
WATER VALVE
CATCH BASIN
CURB INLET
IRON PIN
MANHOLE
TELEPHONE POLE
TRANSFORMER
CONCRETE MONUMENT
GAS METER
WATER METER
FENCE
RAILROAD
BOLLARD
BRIDGE

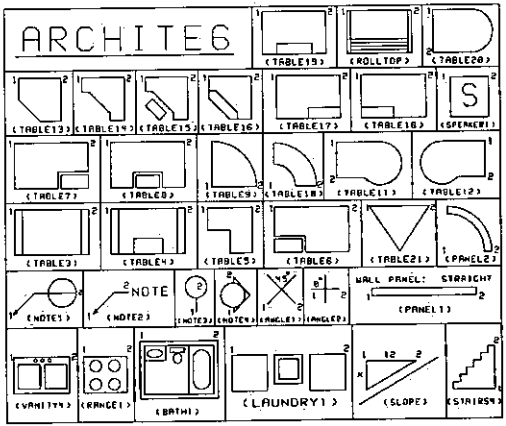
ARCHITE2
FIREPLS
FIREPL6
CHAR1
MUSIC1
PLNTER1
OTTO1
PIANO1
BOOK1
COCKTAIL TABLE
TABLE1
TABLE2
MUSIC CENTER
PLANTER
OTTOMAN
GRAND PIANO
BENCH

ARCHITE4
PLANT1
PLANT2
PLANT3
PLANT4
PLANT5
PLANT6
PLANT7
PLANT8
PLANT9
PLANT10
PLANT11
PLANT12
PLANT13
PLANT14


ARCHITE7
REFERENCE
(REF1)
REFERENCE
(REF2)
REFERENCE
(REF3)
COLUMN LINE
(COLUMLIN)
LINE NOTE
(NOTE5)
LINE NOTE
(NOTE6)
LINE NOTE
(NOTE7)
EXIT LIGHT
EXIT
(EXIT1)
S
(SPEAKER1)
S
(SPEAKER2)
(GRILL)
ACCESS PANEL
(ACCESS)
EXIT LIGHT
(EXIT2)
DIFFUSER
(DIFFUSE1)
DIFFUSER
(DIFFUSE2)
DIFFUSER
(DIFFUSE3)
DIFFUSER
(DIFFUSE4)
ELEVATION
(ELEV)
STEEL COLUMN
(COLUMN1)
PIPE COLUMN
(COLUMN2)
CONCRETE COLUMN
(COLUMN3)
TUBE COLUMN
(COLUMN4)
RECESSED 2'-2" FIXTURE
(FIX3B)
SURFACE 1'-4" FIXTURE
(FIX1A)
RECESSED 1'-4" FIXTURE
(FIX1B)
SURFACE 2'-4" FIXTURE
(FIX2A)
RECESSED 2'-4" FIXTURE
(FIX2B)
SURFACE 2'-2" FIXTURE
(FIX3A)


ARCHITE5
(SIGN1)
(SIGN2)
(SIGN3)
(SIGN4)
TOPSO CON
(TOPSOIL)
BIKE RACK
(BIKERACK)
BENCH SEAT
(BENCH)
BUS SHELTER
(BUSSHELT)
(STYLE1)
(STYLE2)
(ARROW1)
(LIGHTPOL)
BUS
NORTH
(NORTH1)
(SCALE1)
(SCALE2)
(SCALE3)
(SCALE4)
(SCALE5)
(SCALE6)
(SCALE7)
(SCALE8)

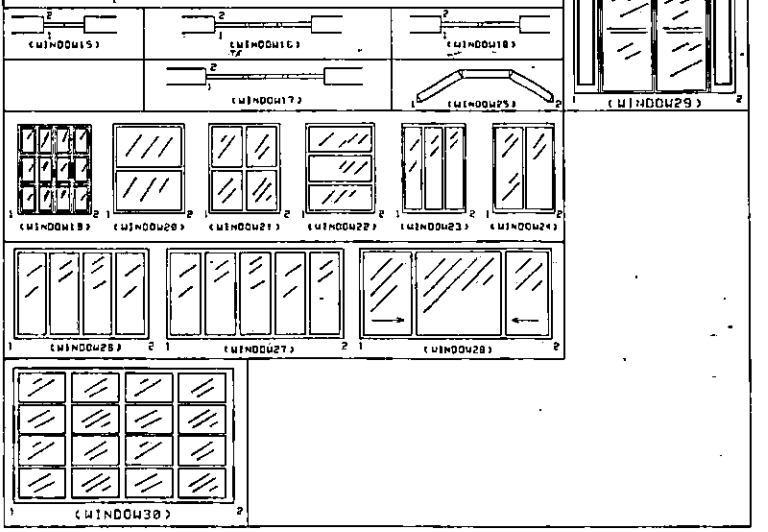

ARCHITE8
ROOM NUMBER
(ROOMNO1)
DOOR NUMBER
(DOORNO1)
COUNTER LAVATORY
(LAV1)
RELEASED FOR CONSTRUCTION
(NOTE8)
FOR BIDDING ONLY
(NOTE9)
FIXTURE: PORCELAIN
(FIX4)
FIXTURE: CAN
(FIX5)
FIXTURE: CAN DIRECTURAL
(FIX6)
FIXTURE: CAN EYEBALL
(FIX7)
FIXTURE: BOX
(FIX8)
FIXTURE: WALL MOUNT
(FIX9)
FIXTURE: WALL MOUNT
(FIX10)
LINE BREAK
(LINE15)
MARKER: REVISION
(MARKER1)
MARKER: DETAIL
(MARKER2)
TANK
(TOILET2)
WALL HUNG FLUSH VALVE
(TOILET3)
FLOOR MOUNT FLUSH VALVE
(TOILET4)
URINAL
(TOILET5)
TANK
(TOILET6)
WALL FLUSH VALVE
(TOILET7)
NORTH
(NORTH3)
MAN
(MAN1)
WOMAN
(MAN2)
WOMAN
(WOMAN1)

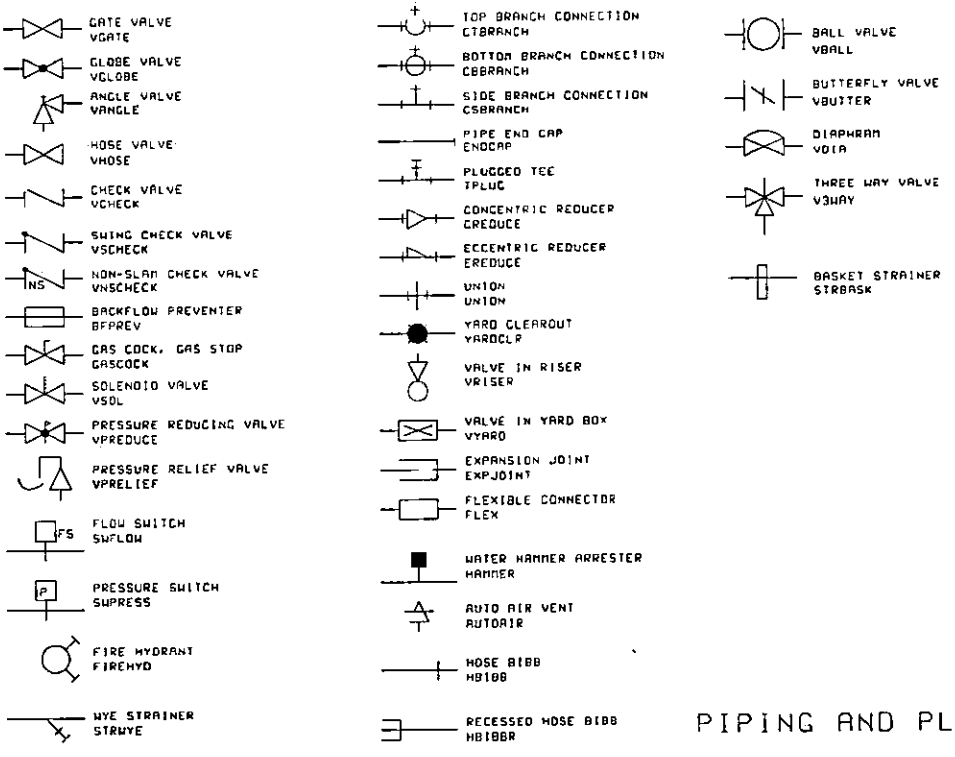

ARCHITE6
(TABLE19)
(ROLLTOP)
(TABLE20)
(TABLE13)
(TABLE14)
(TABLE15)
(TABLE16)
(TABLE17)
(TABLE18)
S
(SPEAKER1)
(TABLE7)
(TABLE8)
(TABLE9)
(TABLE10)
(TABLE11)
(TABLE12)
(TABLE3)
(TABLE4)
(TABLE5)
(TABLE6)
(TABLE21)
(PANEL2)
(NOTE1)
NOTE
(NOTE2)
(NOTE3)
(NOTE4)
(ANGLE1)
(ANGLE2)
WALL PANEL: STRAIGHT
(PANEL1)
(VANITY1)
(RANGE1)
(BATH1)
(LAUNDRY1)
(SLOPE)
(STAIRS1)

FURNITU1

24×60 (DESK1)	30×45 (DESK2)	30×55 (DESK3)	30×60 (DESK4)	30×65 (DESK5)	30×70 (DESK6)	36×24 (DESK7)	36×48 (DESK8)
36×60 (DESK9)	36×70 (DESK10)	36×72 (DESK11)	36×78 (DESK12)	36×72 (DESK13)	48×120 (DESK14)	48×72 (DESK15)	48×96 (DESK16)
ROUND TABLE (TABLE10)	20×72 (TABLE11)	24×60 (TABLE12)	30×45 (TABLE13)	30×55 (TABLE14)	30×60 (TABLE15)	30×65 (TABLE16)	30×70 (TABLE17)
36×24 (TABLE18)	36×48 (TABLE19)	36×60 (TABLE20)	36×70 (TABLE21)	36×78 (TABLE22)	38×72 (TABLE23)	42×120 (TABLE24)	48×72 (TABLE25)
48×96 (TABLE26)	SQUARE TABLE (TABLE27)	PARTITION 2×18 (PART1T1)	PARTITION 2×24 (PART1T2)	PARTITION 2×30 (PART1T3)	PARTITION 2×36 (PART1T4)	PARTITION 2×42 (PART1T5)	PARTITION 2×48 (PART1T6)
PARTITION BIN 14×30 (PARTBIN1)	PARTITION BIN 14×36 (PARTBIN2)	PARTITION BIN 14×42 (PARTBIN3)	PARTITION BIN 14×48 (PARTBIN4)	PARTITION SHELF 12×30 (PARTSHF1)	PARTITION SHELF 12×36 (PARTSHF2)	PARTITION SHELF 12×42 (PARTSHF3)	PARTITION SHELF 12×48 (PARTSHF4)
18×30 (FILE1)	18×36 (FILE2)	18×42 (FILE3)	CREDENZA 28×60 (CRED1)	CREDENZA 28×72 (CRED2)	CHAIR (CHAR2)	CHAIR (CHAR3)	

WINDOWS1





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